MINERALS

ECONOMIC & REGIONAL DEVELOPMENT AGREEMENT

Final Report
TASK 4 - Geotechnical Database
for Ground Control
Numerical Modelling
BEAP USER MANUAL

by

Inco Limited, Thompson Division
Thompson, Manitoba

March 1990

Canada

Manitoba

MG-13097A

CANADA - MANITOBA

MINERAL DEVELOPMENT AGREEMENT

RESEARCH & DEVELOPMENT IN MANITOBA MINES

Work on the project was conducted under the auspices of Canada Centre for Mineral and Energy Technology, Energy, Mines and Resources Canada under the Canada/Manitoba Mineral Development Agreement.

DSS File NO. 14SQ.23440-4-9147-4
DSS Contract Serial No. OSQ84.00404
CANMET Project NO. 4-9147-4
CANMET Scientific Authority: Y.S. Yu

Final Report
TASK 4 - Geotechnical Database
for Ground Control
Numerical Modelling
BEAP USER MANUAL

by

Inco Limited, Thompson Division
Thompson, Manitoba

March 1990

75pp

MICROMEDIA

CANADA - MANITOBA

MINERAL DEVELOPMENT AGREEMENT

RESEARCH & DEVELOPMENT IN MANITOBA MINES

GEOTECHNICAL DATABASE FOR GROUND CONTROL

Numerical Modelling

Beap User Manual

This has been prepared as part of the

TASK 4
Geotechnical Database for Ground Control
of the
Canada/Manitoba Mineral Development Agreement

by

J.A.C. (Tony) Diering
Steffen, Robertson, & Kirsten (B.C.) Inc.
Sub-contracting to
Inco Limited, Thompson Division, Manitoba

DSS File No.: 14SQ.23440-4-9147-4
CANMET Project No.: 4-9147-4
CANMET Scientific Authority: Y.S. Yu

90-05659/1

USER DOCUMENTATION

FOR

BEAP : VERSION 2.0

BOUNDARY ELEMENT ANALYSIS PACKAGE

BOUNDARY ELEMENT MODULE

VOLUME 1 OF 1 : USER MANUAL

REF 54703 JANUARY 1990

TABLE OF CONTENTS

		Page
1.0	OVERVIEW	1
1.1	APPLICATION AREAS	1
1.2	FEATURES	1
1.3	LIMITATIONS	2
1.4	ABOUT THIS MANUAL	2
1.5	A QUICK EXAMPLE	2
1.6	NON-CONFORMING ELEMENTS	5
1.7	GEOMETRIC NODES AND FUNCTION NODES	5
2	INSTALLATION	8
3	PREPARATION OF USER INPUT FILES	17
3.1	FILE.1 -- Global nodes and element definitions	17
3.2	FILE.2 -- Element code and subregion definitions	21
3.3	FILE.3 -- Remaining input data	23
3.4	Keywords	24
3.5	An example of FILE.3	41
4	PROGRAM MODULES WITHIN BEAP	42
5	RUNNING BEAP	55
6	REFERENCES	57
	APPENDIX 1 -- Sample data files	58

LIST OF FIGURES

Figure		Page
1.5.1	Example of multiple excavation problem.	3
1.5.2	Boundary element mesh for multiple excavation problem	4
1.6.1	Different types of boundary element.	6
1.7.1	Geometric nodes for a single element	7
1.7.2	Function nodes for a single element	8
2.4.1	Geometry for solid cube test example	12
2.4.2	Geometry for spherical excavation test problem	13
2.4.3	Geometry for cube excavation test problem	14
2.4.4	Five interacting excavations	15
2.4.5	Two bonded cubes test problem.	16
3.1.1	From physical model to numerical model	20

1.0 AN OVERVIEW

BEAP facilitates fully three-dimensional stress analysis of underground excavations. It is an interactive package which has been designed and written specifically for application to mining problems. Excavation surfaces, the earth's surface and geological interfaces are all modelled with boundary elements. The element type used is a non-conforming quadratic variation element which provides numerous advantages for the stress modelling of mining excavations. The package evolved from the boundary element package MBEM which already has been used or installed in many countries.

1.1 APPLICATION AREAS

- Analysis of complex service excavations such as tunnel intersections and shaft layouts
- Analysis of excavations created by mass mining operations, including open stoping, block caving VBM or VCR and various cut and fill methods.
- Analysis of drawpoint layouts
- Pillar design
- Interaction of open pit and underground excavations

1.2 FEATURES

- IBM-PC/AT or compatibles
- Simple data preparation
- Designed specifically for rock mechanics problems
- Uses an advanced element type known as a non-conforming quadratic element
- Enables geometries with planar or curved surfaces to be modelled
- Capacity of up to 256 primary elements (2300 function nodes) or 6912 degrees of freedom
- Primitive stress loading or gravity loading
- Named elements and benchmarks for easy user identification
- Use of iterative solution technique and lumping to reduce computing requirements
- Modelling of different geological regions or a linear elastic backfill (Up to 5 subregions have been modelled successfully)
- Use of repeatability for problems with repeatable geometry or use of symmetry
- Robust against user abuse with various re-start options
- Stresses and displacements available within the rock mass or at excavation surfaces

- Subregions may be finite or infinite. Material is linear elastic and isotropic.

- Fully compatible with BEAP-DD and BEAP-M

1.3 LIMITATIONS

- Tabular excavations with length to height ratios exceeding 50 are not easily modelled. Such excavations are better modelled with displacement discontinuity elements (such as found in BEAP-DD for example).

- Complex non-homogeneous geologies cannot be modelled. Whilst two and even three or four subregion problems have been modelled, caution and experience are needed before attempting to use multi subregion capabilities within the program.

- Large problems may require long execution times. On IBM-PC type equipment (even using 80386 processors), overnight runs will usually be required.

This manual has been prepared for the assistance and guidance of users of BEAP. Whilst care has been taken in the presentation of this manual, the user should treat results with due caution and engineering judgement. By the very nature of such programs, an in-experienced user can easily obtain erroneous results. Moreover, as there so many different types of problems which can potentially be analyzed by such a program, it is possible (and more likely probable that there will be program errors in some circumstances). Such errors can only be ironed after continued use and debugging of the program.

1.4 ABOUT THIS MANUAL

This manual assumes that the user is already familiar with the concepts of three-dimensional stress analysis, and the associated concepts of working with a three-dimensional geometry. For further information in this regard, the user is referred to the list of references listed at the end of the manual.

Section 2 of this manual describes installation procedures, while Section 3 describes the preparation of the input data files. Section 4 describes each of the program modules in BEAP. Less experienced users are well advised to read through the papers listed as Supplementary Reading in Section 6.

1.5 A QUICK EXAMPLE

Figure 1.5.1 shows a problem which was successfully modelled in three dimensions. Four mining steps were simulated representing different stages of extraction for an approximately tabular ore body (10 to 35 m thick) dipping at about 70 degrees. One of the objectives of this modelling exercise was to assess stress changes taking place at the shaft locations (see Figure 1.5.1).

The boundary element mesh for Step 2 is shown in Figure 1.5.2. This example contains about 160 boundary elements for modelling 8 excavations. Although the description of this example is "quick", running times on an IBM-AT (8 MHtz) would be around 30 hours.

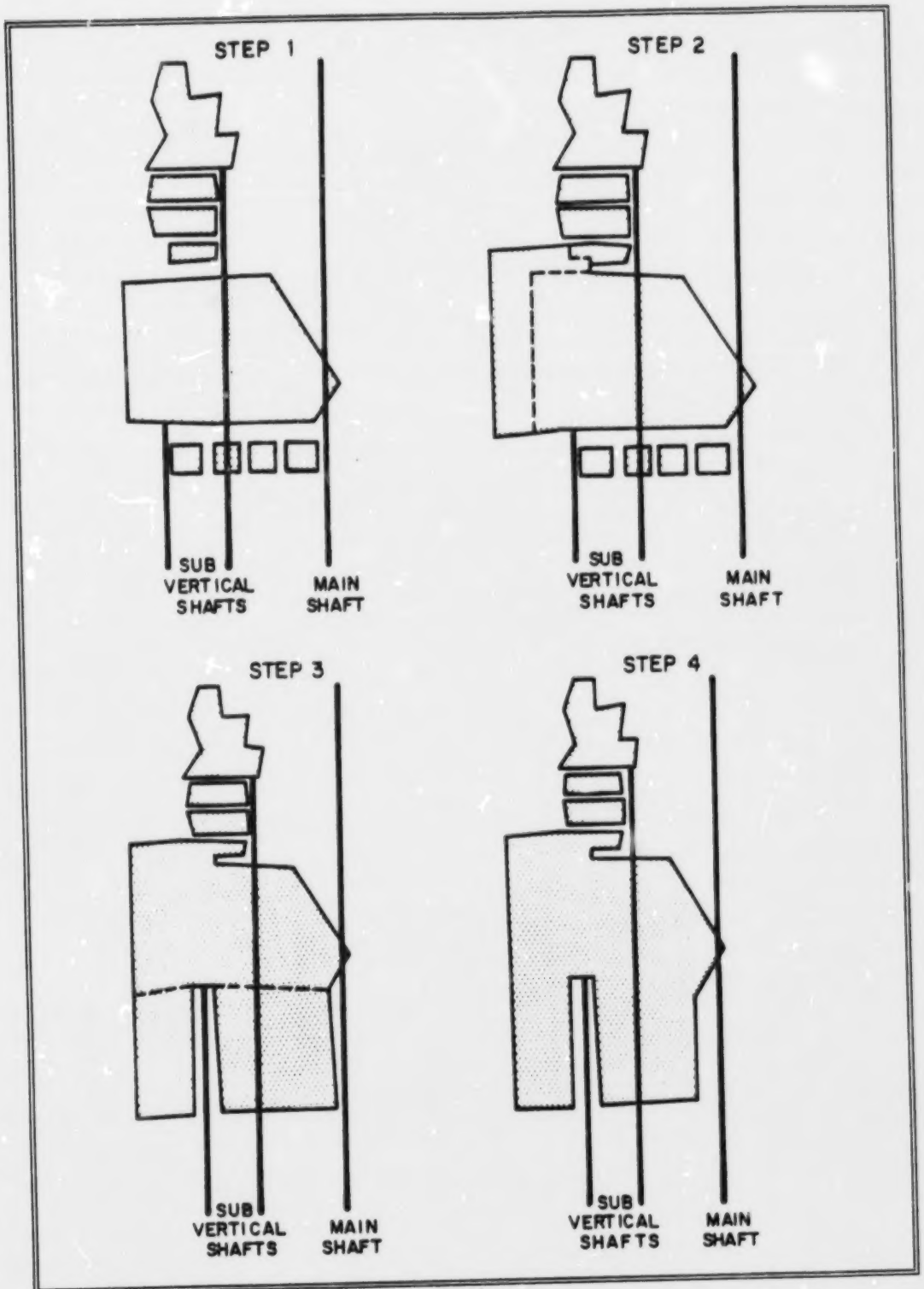

Figure 1.5.1: Example of Multiple Excavation Problem

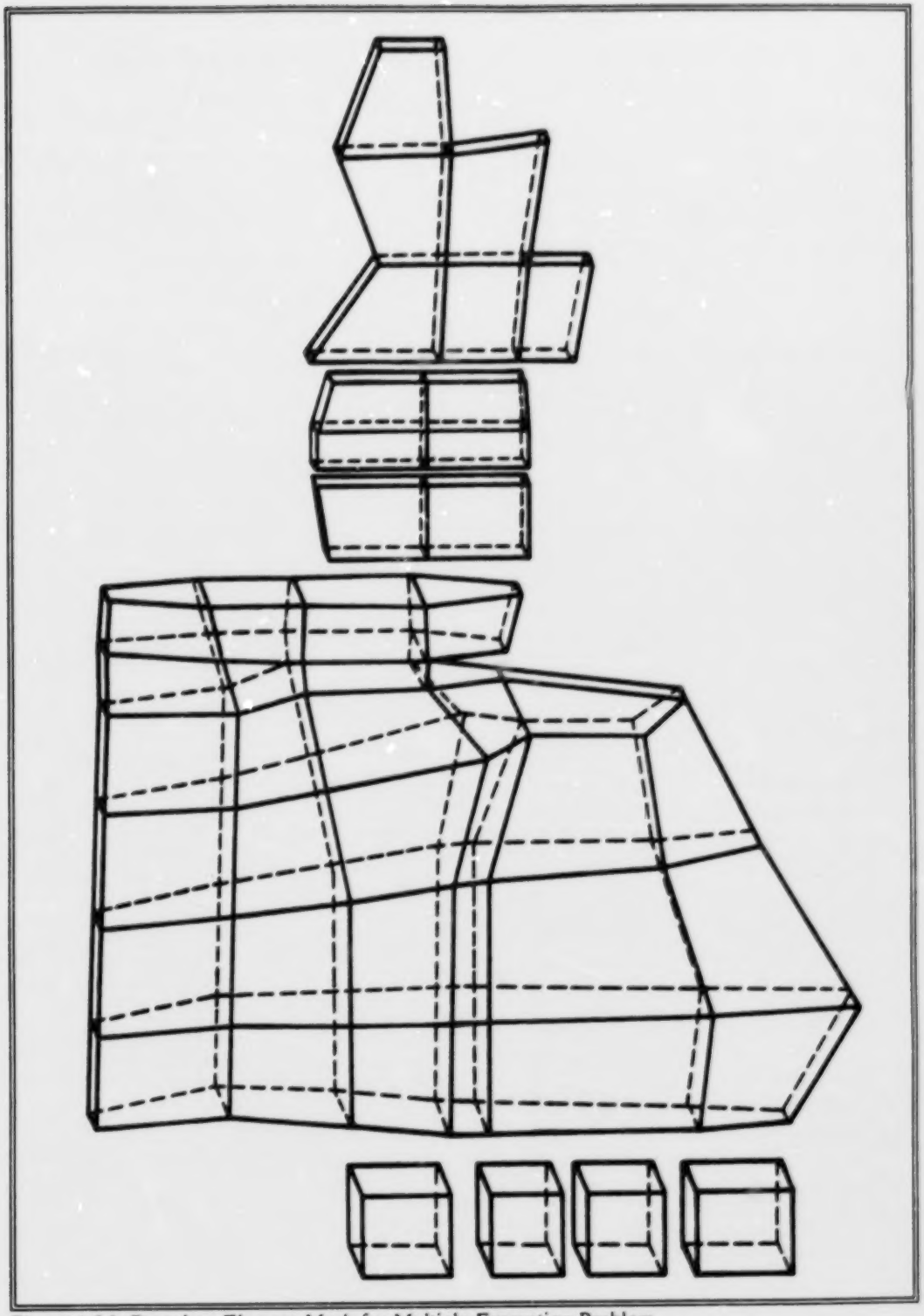

Figure 1.5.2: Boundary Element Mesh for Multiple Excavation Problem

1.6 NON-CONFORMING ELEMENTS

Program BEAP has evolved (via a long and devious route) from its predecessor MBEM. MBEM employed many simple elements to model an excavation. BEAP makes use of much fewer elements, each of which are more powerful. These elements are referred to as non-conforming elements, mainly because the interaction between adjacent elements is of a discontinuous nature. Examples of different element types are given in Figure 1.6.1.

1.7 GEOMETRIC NODES AND FUNCTION NODES

Nodes are points in space. A node is uniquely defined by a coordinate location (x, y, z). Elements contain nodes. The location in space and size and shape of the element depend upon the location of the nodes defining that element. The non-conforming elements used by BEAP makes use of two types of nodes: geometric nodes and function nodes (see Figure 1.7.1).

Geometric nodes are used to define the location of the element in space. 3, 4, 8 or 9 geometric nodes are required for each element depending upon whether the element is triangular, quadrilateral, planar or curved. The relative locations of these geometric nodes are shown in Figure 1.7.1 (top). Once these geometric nodes have been used to define the element locations, the program then generates and uses "function" nodes located within each element for subsequent processing. These function nodes are defined in terms of a local or intrinsic coordinate system for each element.

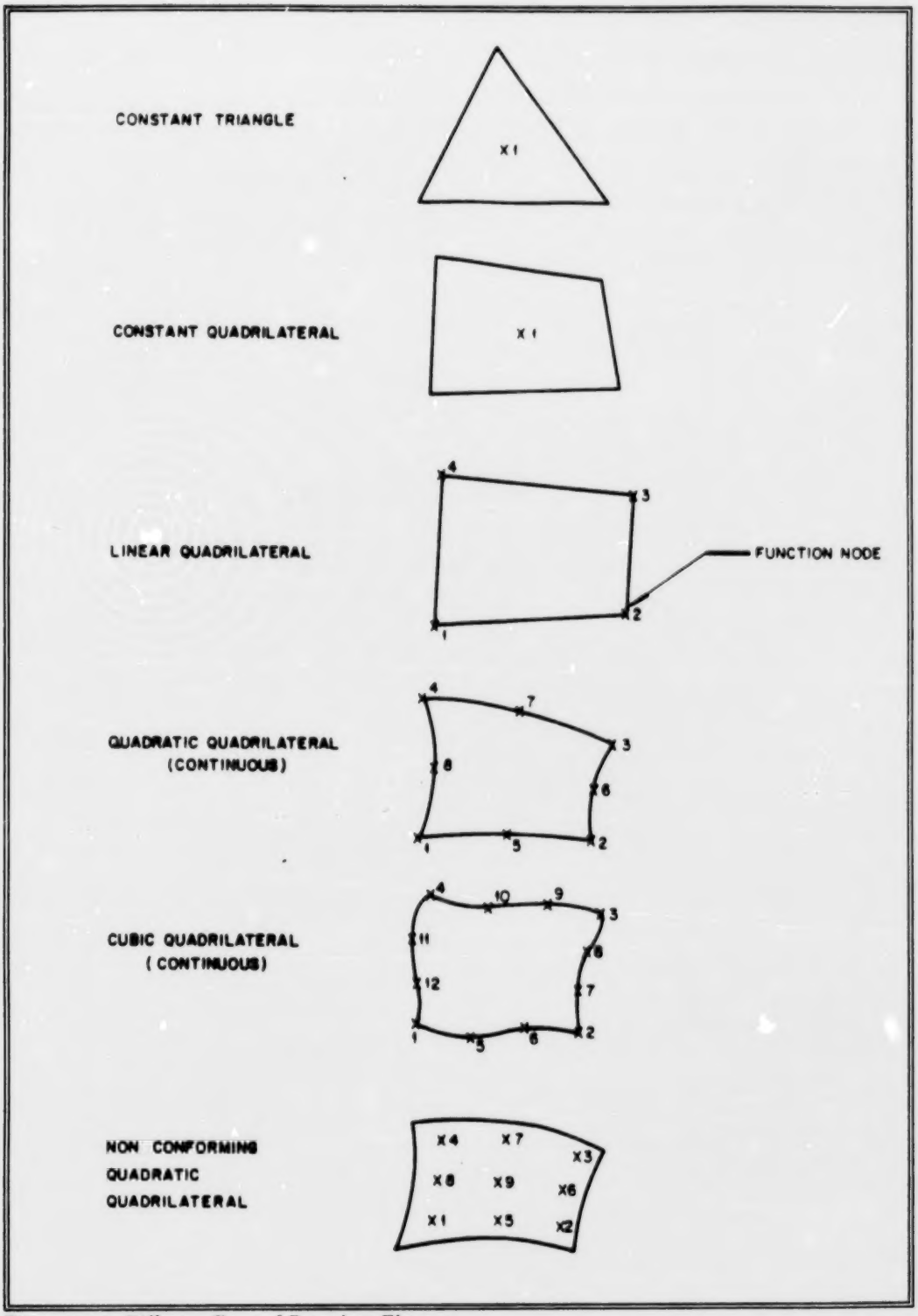

Figure 1.6.1: Different Type of Boundary Element

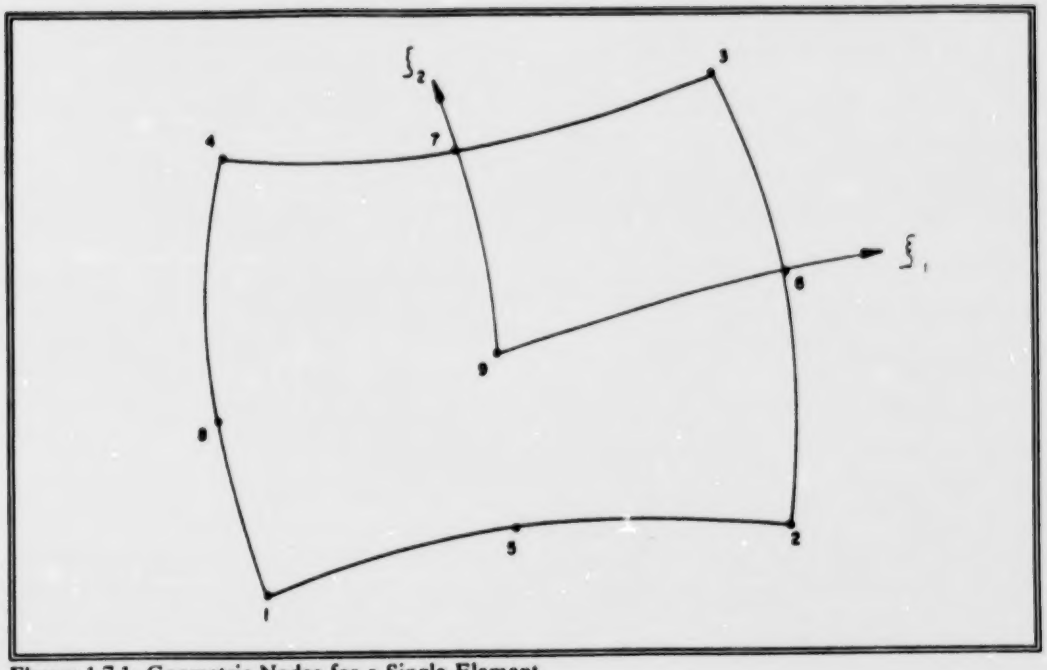

Figure 1.7.1: Geometric Nodes for a Single Element

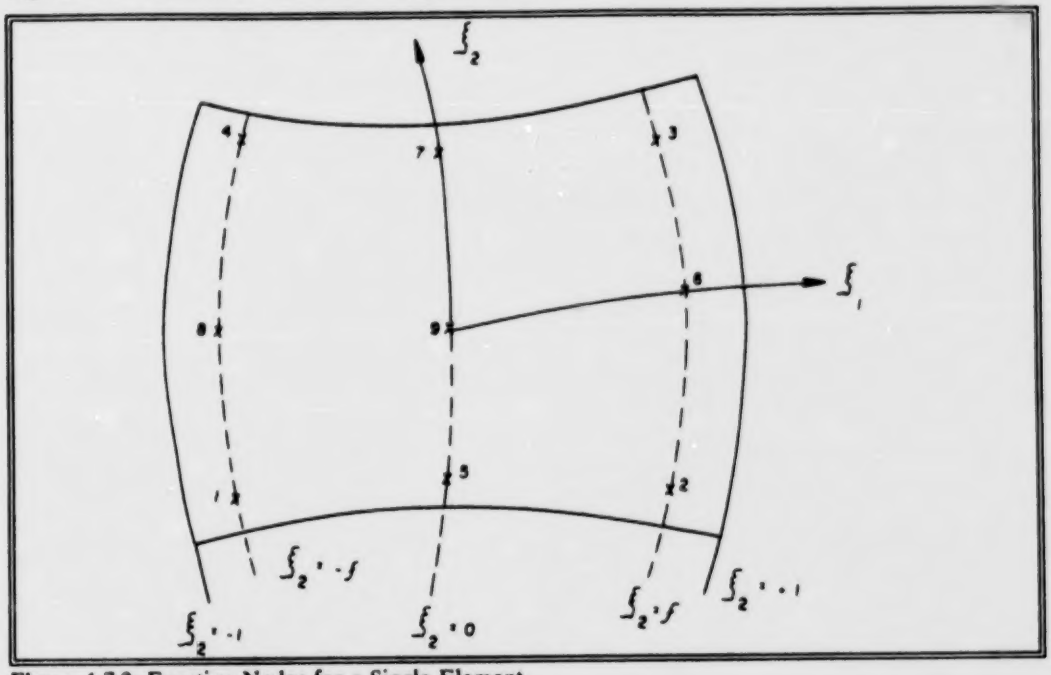

Figure 1.7.2: Function Nodes for a Single Element

2.0 INSTALLATION

This installation procedure applies only to IBM-PC compatible equipment running DOS. BEAP has already also been installed successfully on DATA GENERAL and VAX mini computers.

2.1 GETTING STARTED

It is advisable to create a separate subdirectory which will contain the BEAP program modules and a second subdirectory to contain the data files and database. A sound working knowledge of DOS is required for a successful installation of BEAP. Let us call the program subdirectory 'BEAP' and assume that it is on the C Drive. Let us call the database subdirectory 'BEAPDATA' and assume that it is on the D Drive. If there is insufficient space on the C drive to install all of the executable code for BEAP, then it is still necessary to create a subdirectory C:\BEAP which MUST contain the message files for BEAP (and also BEMPOST, BEAP-DD and BEAP-M if these are also installed). The AUTOEXEC.BAT must be modified so that the BEAP directory which contains the executable code (ie all the .EXE files) is included in the DOS PATH. The CONFIG.SYS file must also be modified to ensure that the 'FILES = 20' statement is included.

The program modules occupy around 3 Megabytes of disk space. The test database supplied with BEAP is very small (less than 1 MByte). However, storage requirements for the database can be very large. At least 15 Megabytes should be available for medium-sized problems with 30 Megabytes or more for larger problems. A problem with around 160 elements will require about 25 Megabytes of disk space.

2.2 DATABASE FILES

The user is required to create three (or sometimes more) input data files. Usually, these should have the file type extensions .1 .2 and .3 for easy separation. When a new database is created using M101, numerous other data files will be created in the current subdirectory from which the program is being run. Thus, it is possible to run the program from a separate sub-directory from that which contains the data input files, although this is usually not necessary.

Most of the database files are in binary code and cannot be usefully displayed by the user. There are program modules for decoding or displaying these files where necessary.

Database files are as follows:

BOTH.BE	COMMON BLOCK
BEMCO.BE	INFLUENCE COEFFICIENTS
BEMMESH.BE	PLOT FILE FOR MESH GEOMETRY
BEMMESHE.BE	PLOT FILE FOR ELEMENT NUMBERS
BEMMESHN.BE	PLOT FILE FOR NODE NUMBERS
BENSTOR.BE	BENCHMARK FILE
BVECT.BE	RIGHT-HAND SIDE VECTOR
LIND.BE	LUMPING MAPS
LUMSTOR.BE	MAIN STORAGE FILE FOR ELEMENTS
LUMSTOR2.BE	SECONDARY STORAGE FILE FOR ELEMENTS
MESAGE.BE	MESSAGE FILE (Should be in C:\BEAP subdirectory)
SELFCO.BE	ELEMENT SELF COEFFICIENTS

2.3 PROGRAM FILES

The following .EXE files make up the BEAP system. Programs may be run independently, or via the menu driver BEAP.EXE.

BEAP Menu driver for BEAP and BEAP-M

M101 Initialize an entire new database (BEAP and BEAP-DD)
M103 Initialize displacements and tractions

M201 Read in node and element geometry
M202 Check nodes and elements (usually included in M203)
M203 Read in element TYPE definitions and check
M204 Print selected elements or nodes (Detailed print)
M205 Display selected elements to screen
M209 Print all element geometry (batch mode of M204)

M301 Prepare a 3-D view of mesh/elements/nodes

M401 Read in other input data
M402 Print header page and selected information

M501 Apply tractions due to primitive stresses
M502 Calculate lumping maps (BE-BE)
M503 Calculate influence coefficients (BE-BE)
M504 Solve equations (BE)
M507 Calculate benchmarks (BE)
M508 Calculate surface stresses and normal displacements
M509 Convert surface stresses into benchmark values

M601 Print selected element results
M602 Print all element results (batch mode of M601)
M603 Print selected benchmark results

M701 Prepare selected element results for plotting
BEMPOST Post processor for benchmarks

M901 Print nodes and element in data file format
M902 As for M901, with reversed element ordering
M903 Display/modify number of nodes, benchmarks and elements
M905 Load in a new message file

2.4 TEST DATA FILES

A number of test data files have been included at the end of this manual to help new users become familiar with BEAP. The data files and examples are as follows:

Unit cube under uniaxial compression:

Data files are BOX.1, BOX.2 and BOX.3. Geometry for this problem is shown in Figure 2.4.1.

Spherical cavity test problem. Data files are SPHERE.1, SPHERE.2 and SPHERE.3. Geometry for this problem is shown in Figure 2.4.2.

Cube excavation test problem:

Data files are CUBEX.1, CUBEX.2, CUBEX.3, CUBEX.BEN and CUBEX.HED. Geometry for this problem is shown in Figure 2.4.3.

Five interacting stopes. Data files for a coarse mesh for this problem are FIVE.1, FIVE.2 and FIVE.3. Geometry for this problem is shown in Figure 2.4.4.

Two bonded cubes test problem:

Data files are INTER.1, INTER.2 and INTER.3. Geometry for this problem is shown in Figure 2.4.5.

Any user running BEAP for the first time should ensure that the program can be made to run with one of these data files (which most closely matches has own problem) before attempting to run BEAP with the actual problem data.

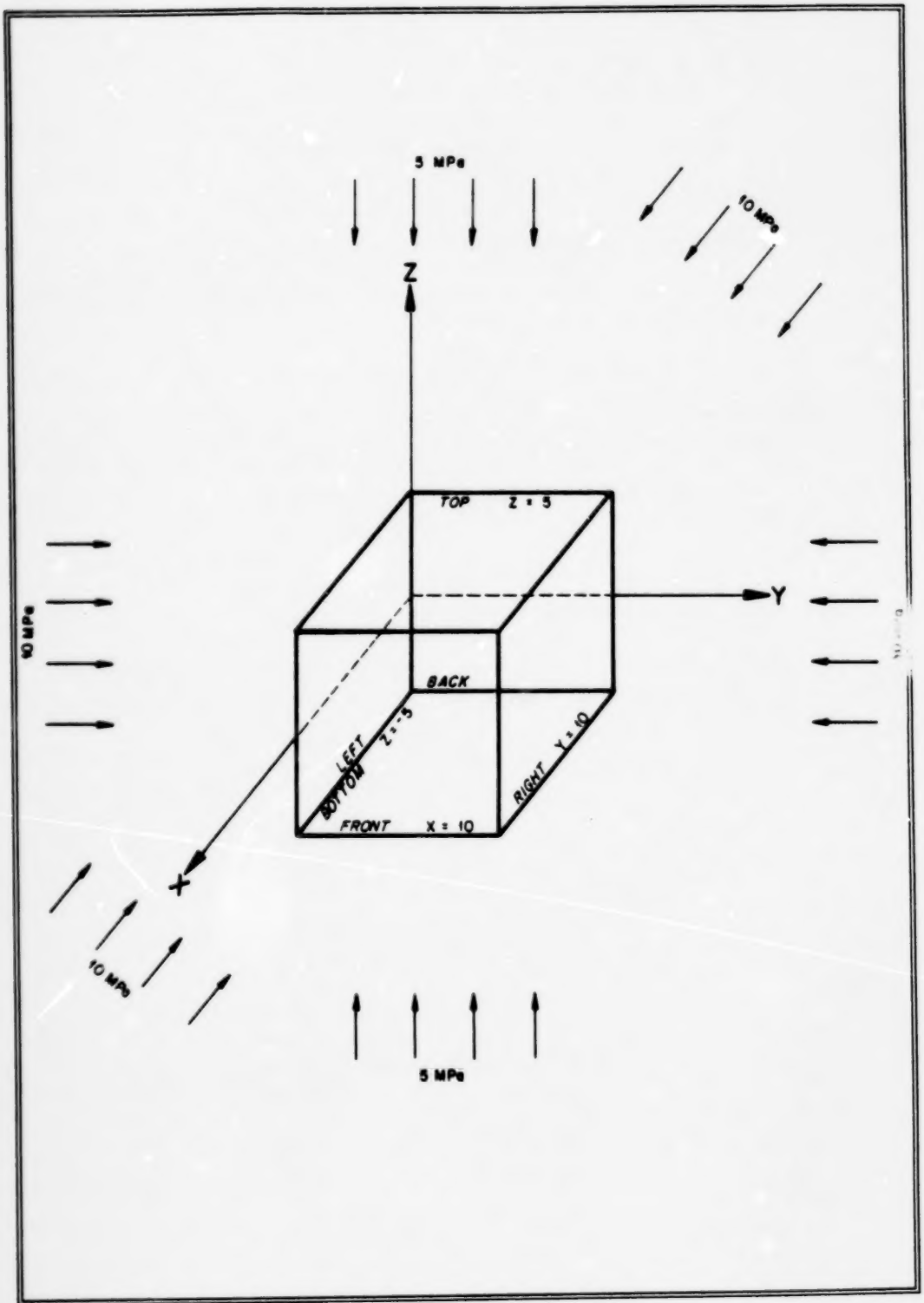

Figure 2.4.1: Geometry for Solid Cube Test Example

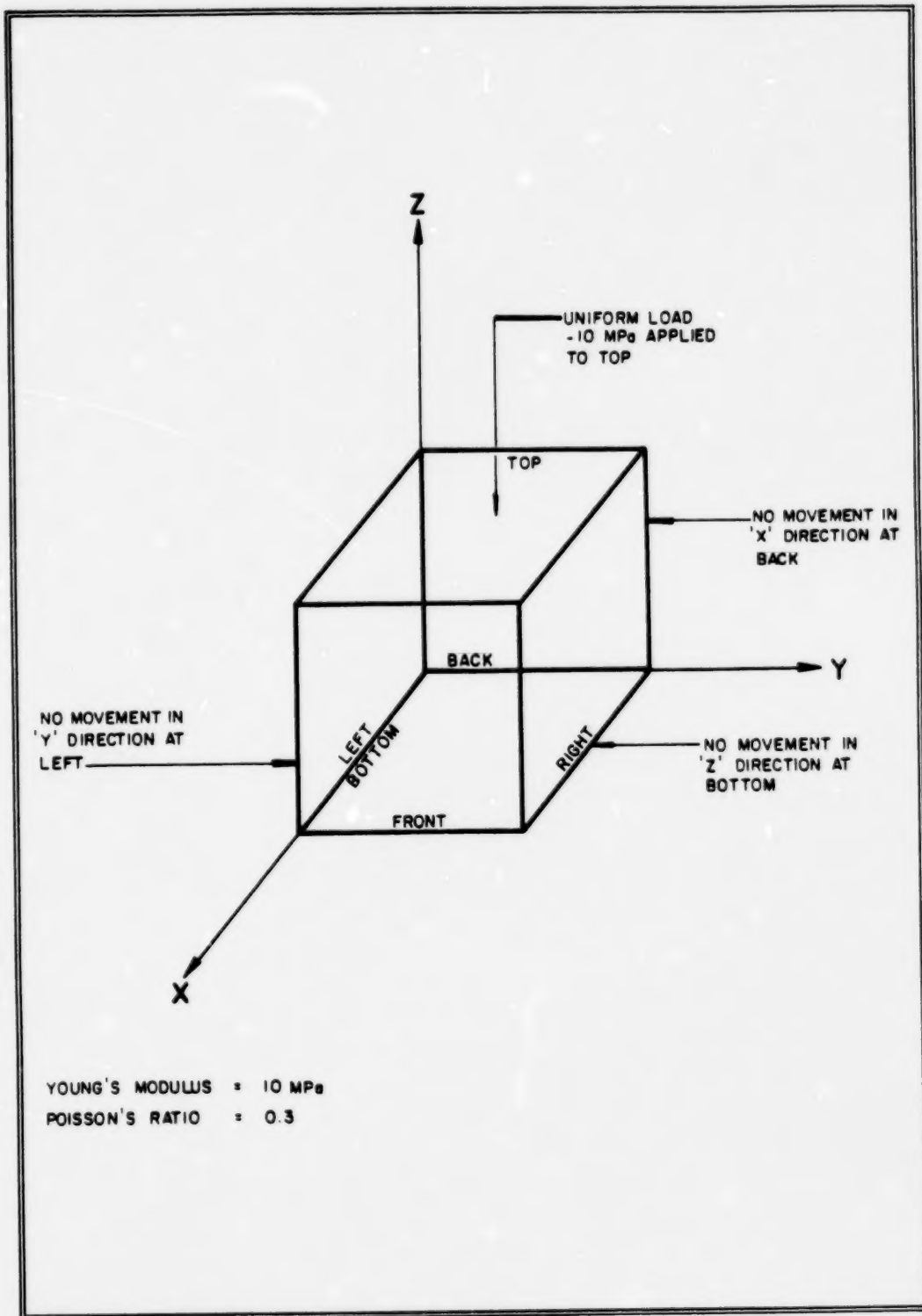

Figure 2.4.2: Geometry for spherical excavation Test Problem

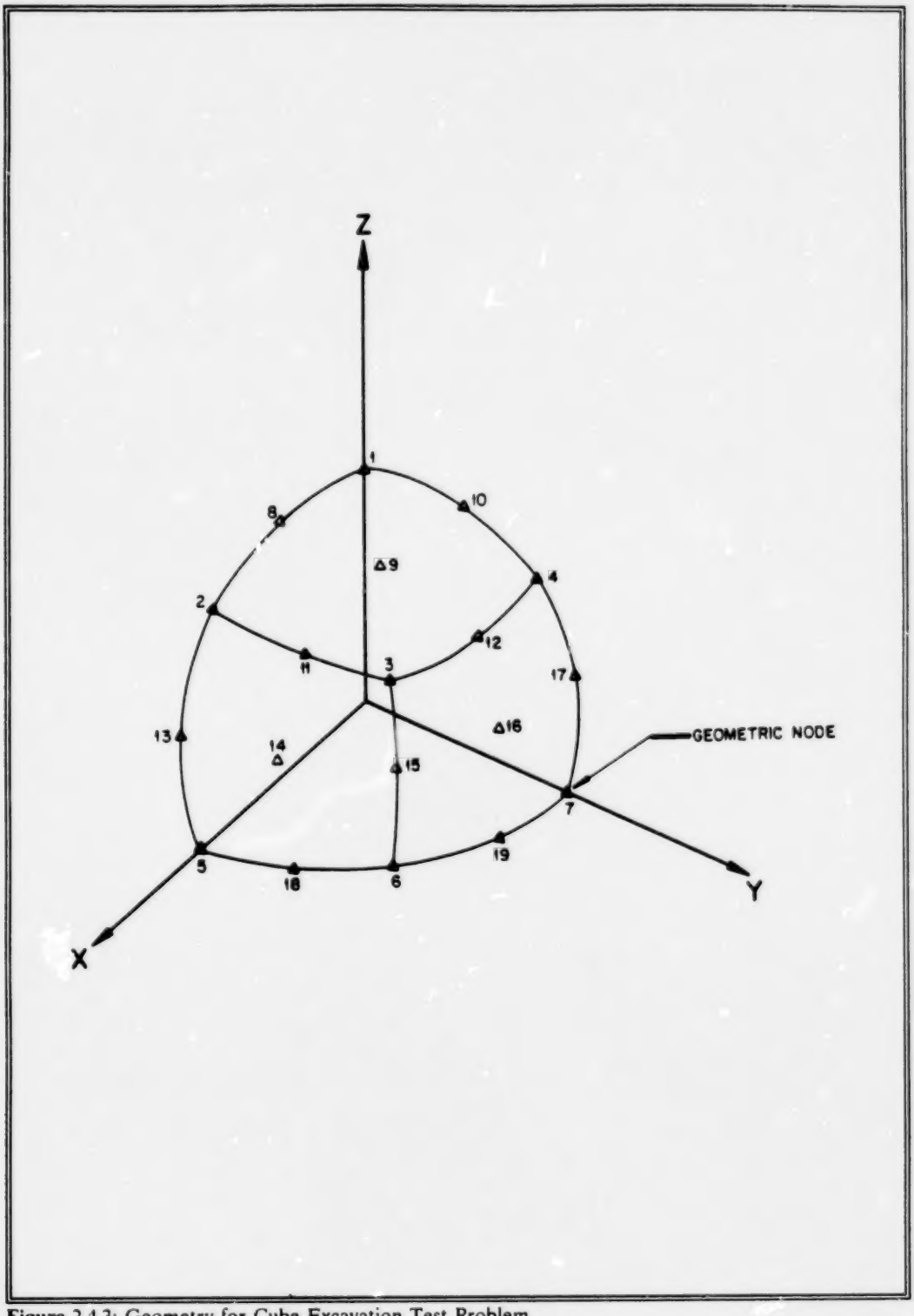

Figure 2.4.3: Geometry for Cube Excavation Test Problem

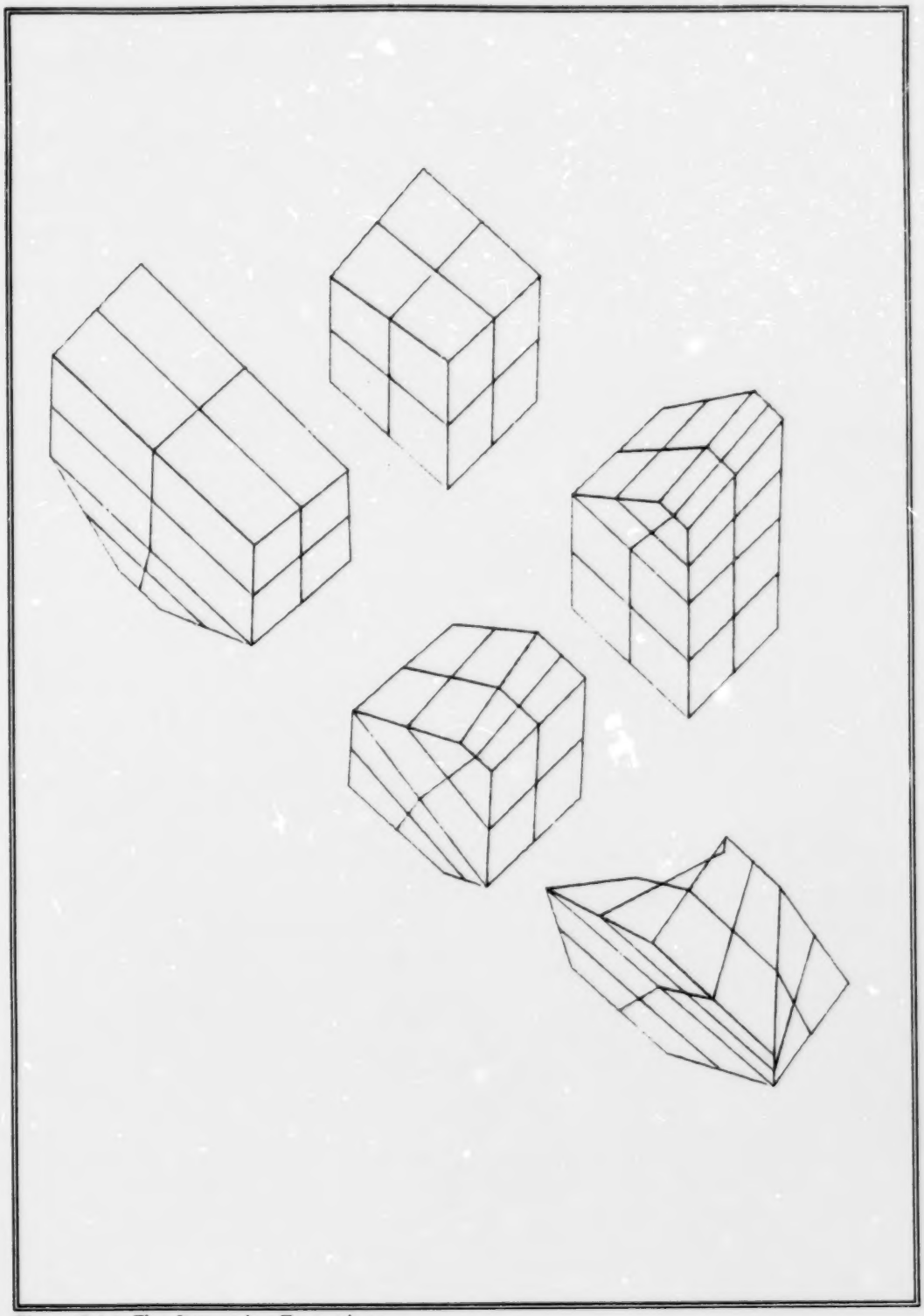

Figure 2.4 4: Five Interacting Excavations

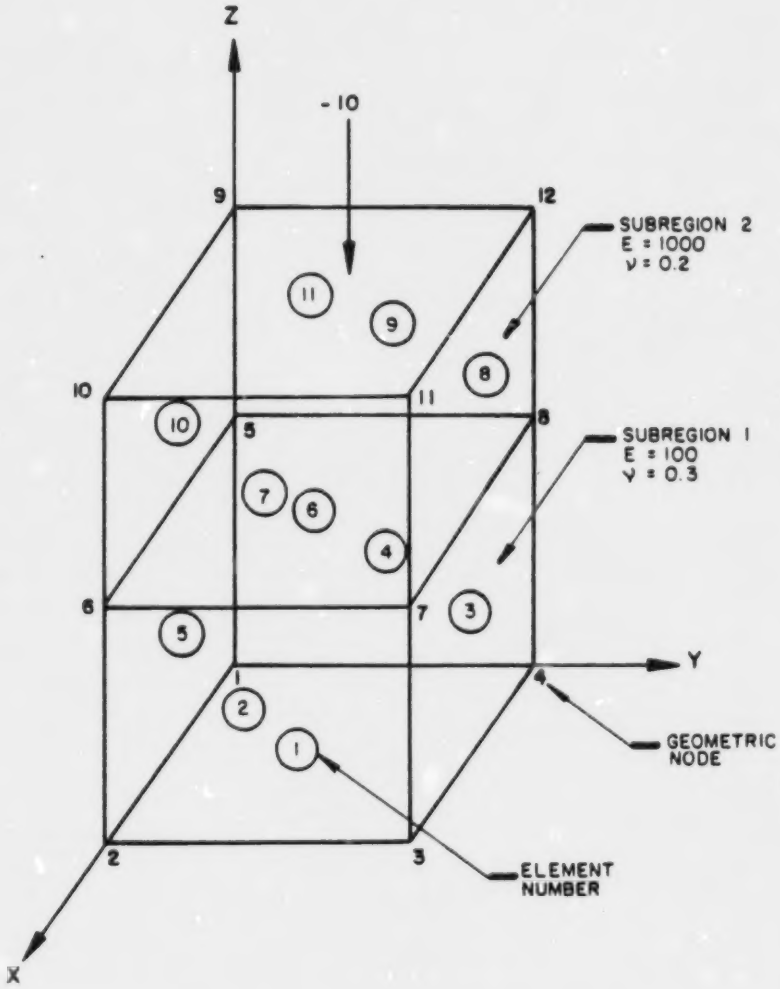

Figure 2.4.5: Two Bonded Cubes Test Problem

3.0 PREPARATION OF USER INPUT FILES

The input data for program BEAP consists of three data files. For the purposes of explanation in the following sections, these files are referred to as FILE.1 , FILE.2 and FILE.3. FILE.1 contains the geometric information for the elements. FILE.2 contains information describing the element boundary condition codes and library code, together with element subregion information. This file is optional in that it is not required for standard mining type problems consisting of 1 subregion. FILE.3 contains information about material properties, primitive stresses, optional symmetry and repeatability and benchmark locations. The next sections detail the structure and format of each data file.

3.1 FILE.1 -- GLOBAL NODES AND ELEMENT DEFINITIONS

This file contains 5 different types of line as follows

Line 1 NOD,NLUM

Line 2 XOF,YOF,ZOF

Line 3 N,X,Y,Z

Line 4a I,M,I1,I2,I3,I4,I5,I6,I7,I8,I9

Line 4b DESCRIP

where :

NOD is the number of global nodes to be defined. This is equal to the number of lines 3 which follow. Max value for NOD is 400.

NLUM is the number of elements to be defined. This is equal to the number of line pairs (4a and 4b) which follow. Max value is 256.

XOF,YOF,ZOF are coordinate offsets used when global nodes are remote from the coordinate origin. Usually use 0,0,0 here.

N is a node number (between 1 and 400)

X,Y,Z are the x,y,z coordinates of that node

I is a element number (between 1 and 256)

M is the number of nodes for that element. Values of 3,4,8 or 9 are recommended for triangles, quadrilaterals and curved quadrilaterals respectively. If M=3, then it is not necessary to define I4,I5,I6,I7,I8 and I9. If M=4, then it is not necessary to define I5,I6,I7,I8 and I9.

DESCRIP is a description of the element consisting of up to 16 characters. This description is purely for user convenience, helping identify elements.

Note

1 Usually, only four nodes will be used to define an element. When defining an excavation, the sense of node numbering is such that if one "stands" inside the excavation, then the nodes are specified in anti-clockwise manner. If one views an element through solid material, then the sense is clockwise.

2. If curved elements are required, then all 9 geometric nodes should be defined for those elements which are curved.

3. It is not necessary for all element numbers to defined. For example it is possible to allocate element numbers 1-20 for the first excavation, 30-50 for the second and so on. In this case, elements 21-29 would be undefined, but this is not a problem, so long the correct definitions are also contained in FILE.2 below.

4. It is also not necessary to define all geometric nodes. ie missing node numbers are not a problem, provided they are not used to define elements.

5. Two elements can border on a single other element, but this should be avoided when curved elements are being used, to avoid creating holes or overlaps in the modelled surface.

6. Figure 3.1.1 shows the logical progression from a formulated problem to the stage of having a number of elements and surrounding nodes.

7. Although there is no logical limit on the area ratio of the largest to smallest element in a mesh, problems may be encountered when very small elements are used along with very large elements. Area ratios of 500:1 should still be acceptable (provided these elements are not close or adjacent to one another). The aspect ratio (in terms of area) for adjacent elements should usually not exceed around 10:1. This is a guideline based upon experience and common sense rather than a rigorous mathematical proof.

An example of this data file (FILE.1) follows : This files defines nodes and elements for a cube shaped excavation.

8,6	Line 1
0,0,0	Line 2
1,0,0,-5	Line 3
2,10,0,-5	
3,10,10,-5	
4,0,10,-5	
5,0,0,5	...
6,10,0,5	
7,10,10,5	
8,0,10,5	Line 3
1,4,1,2,3,4	Line 4
BOTTOM	Line 4b
2,4,3,2,6,7	Line 4a
FRONT	Line 4b
3,4,3,7,8,4	Line 4a
RIGHT	Line 4b
4,4,5,1,4,8	etc
BACK	
5,4,2,1,5,6	...
LEFT	
6,4,7,6,5,8	
TOP	Line 4b

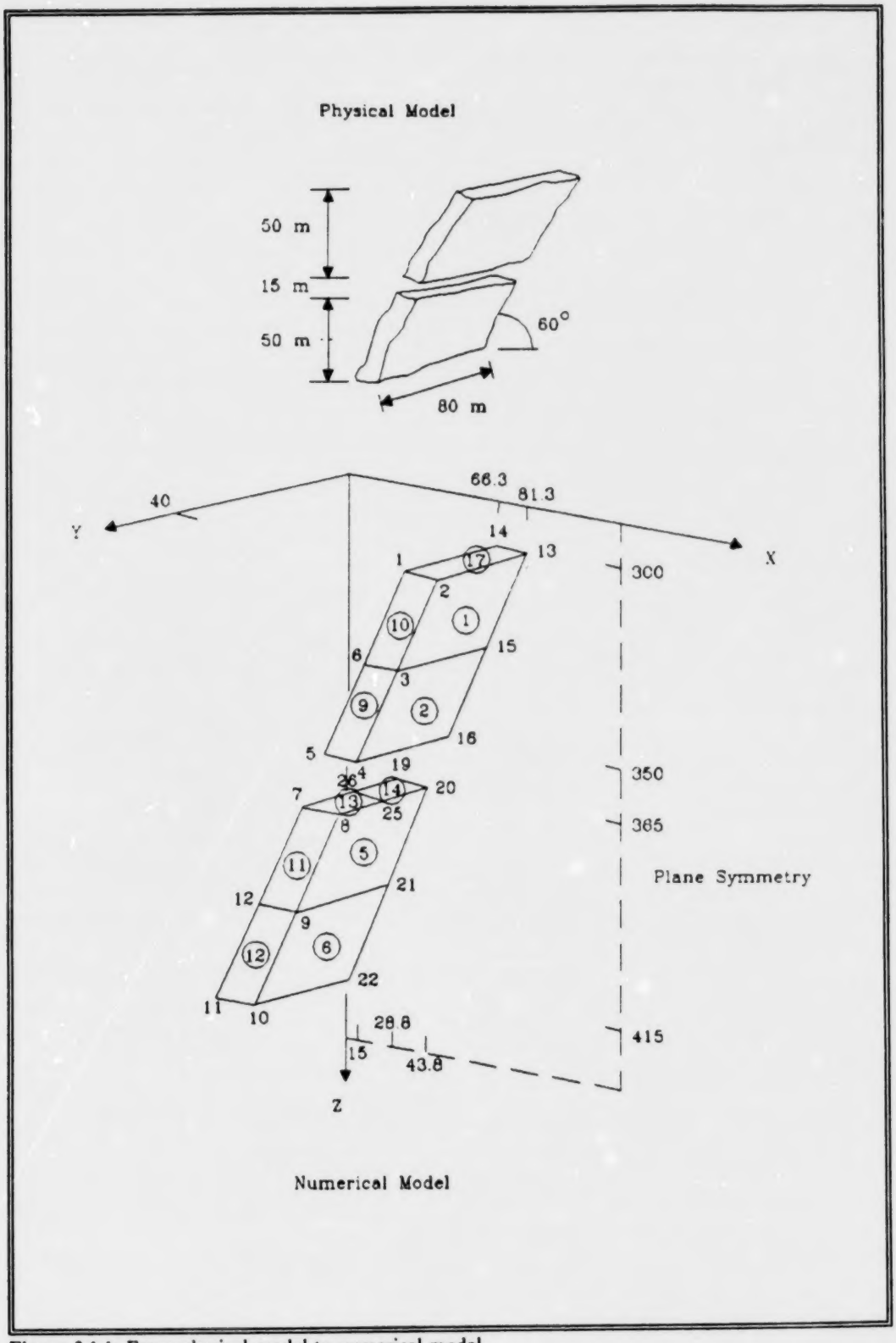

Figure 3.1.1: From physical model to numerical model

3.2 FILE.2 -- ELEMENT CODE AND SUBREGION DEFINITIONS

This file is optional for simple or standard mining problems. It is used to define the following items for selected elements if some or all of these items differ from the pre-allocated default values.

Variable	Description	Default
IEX	Excavation number for user convenience	1
IKOD	Boundary condition code (1 = mined)	1
IPEN	Pen number for plotting the element	1
IMOD	Subregion number containing the element	1
KFACE	Subregion number to which element interfaces	1
LIBKOD	Library code for the element (use 2 usually)	1
ISP1	Spare variable -- use 0	0
ISP2	Spare variable -- use 0	0

The format each line of this data file is the same and is as follows

LB,LE,IEX,IKOD,IPEN,IMOD,KFACE,LIBKOD,ISP1,ISP2

where

LB is the first element number of a series of elements.

LE is the last element number of a series of elements.

Other variables are as defined above.

The last line of this file is

0,0,0,0,0,0,0,0,0,0

An example of this data file for the example of a cube shaped excavation is

1,6,1,1,1,1,0,2,0,0	All six elements have same values
0,0,0,0,0,0,0,0,0,0	End of file marker

Notes

1 The user is referred to the BOUNDR keyword for file FILE.3 for more information on the use of IKOD above.

2. The value of IPEN should have values from 1 to 4 usually.

3. The user is referred to additional notes on the use of subregions for more information on IMOD and IFACE.

4. Using a value of LIBKOD = 1 will use elements for the which the function nodes are located at the intrinsic coordinate locations of +- 0.666667. Using a value of LIBKOD = 2 is generally better and locates the function nodes at intrinsic coordinates of +- 0.8.

5. Defining elements ...e which have been omitted from FILE.1 will cause problems and numerous warnings.

SUBREGION MODELLING

A few pointers for modelling subregions are included here for those users brave enough to try using subregions.

- It is recommended to use a second subregion only if there are two or more geological zones of very different material properties.
- It is not recommended to use a subregion to model backfill at this stage.
- If two subregions are modelled, then the interface elements defining the subregion must be complete. ie The subregion must not be clipped arbitrarily in space remote from the excavations. (Rather use finite subregions enclosed in a box).
- The elements at the subregion must defined as belonging to the softer subregion. The interface subregion will be the harder subregion.
- The ITERAT keyword should be used. Recommend 100 iterations, convergence ratio = 0.01 or 0.05, relaxation factor = 0.4, subiterations = 1 and acceleration factor = 1.0
- It is not possible to specify special or primitive tractions at a subregion interface.
- Subregions do work if the above 'rules' are adhered to. Examples which have been modelled are 2 and 3 bonded cubes, a tunnel with a concrete plug subjected to a water pressure, and a 4 stope problem in which the roof of 3 excavations was in the softer (upper subregion) whilst the sides and fourth excavation were in the lower and harder subregion.

3.3 FILE.3 -- REMAINING INPUT DATA

This third data file looks after the remaining input data. It caters for a wide variety of data types and these data types are specified using keywords. The order in which keywords is specified not critical, but once a keyword has been specified, the structure and format of data which follows is specific to that keyword

Often, the user will prepare several different data files making use of these keywords. For example, it is possible to have one such file whose sole function is to change the number of decimal places used for the generation of plot files, and a second such file which will change the integration accuracy parameters and so on.

Valid keywords are as follows

BENCH	Define benchmark locations
BOUNDR	Define element boundary codes as different from 1
END	Define the end of file
HEADING	Description of the run
INTEGR	Integration data for calculation of influence coefficients
ISSW	Set integer control switches
ITERAT	Iteration information for solution of equations
PRIM0	Constant primitive stresses
PRIM1	Overburden (or depth dependent) primitive stresses
REPEAT	Define offsets for repeatability. This feature is still very much at the research stage, and it's use should be carefully checked.
SPTRAC	Apply special tractions at selected elements/nodes
SSW	Set real control switches
SUBS	Define material properties for each subregion
SYM	Define planes of symmetry as different from 0,0,0

For a 'standard' mining problem with 1 subregion, no symmetry, no repeatability and no special tractions, one would use the following keywords

HEADING
PRIM0
PRIM1
BENCH
SUBS
END

Each keyword is explained separately in the following sections

3.4 KEYWORDS

Keyword **BENCH**

Purpose : To define benchmark locations

Format :

IDB,XBEN,YBEN,ZBEN,ISUB
NAMEB

Variables:

IDB is an integer identifier. Any value between 1 and 9999 may be used for user convenience

XBEN,YBEN,ZBEN are the coordinates of the point

ISUB is the subregion number containing the point. If only 1 subregion has been defined, it is not necessary to specify this variable. ie The format of line 1 above becomes

IDB,XBEN,YBEN,ZBEN

NAMEB is a name for the benchmark location for user convenience. It consists of up to 16 characters.

Notes :

1 The two lines above are repeated for each benchmark point

2 The input of benchmarks is terminated by the following line

-99,0,0,0,0

This line is not followed by a NAMEB line. It is followed by the next keyword

3 There is no fixed limit on the number of benchmarks which may be defined. The only limitations are imposed by disk space, volume of printout and CPU time for benchmark calculations.

4. No facilities are provided for the automatic generation of benchmark locations for BEAP, since this promotes the use of more benchmarks than are often required.

Keyword **BOUNDR**

Purpose : To set special boundary conditions to elements

Format :

LB,LE,LBC
0,0,0

Variables:

LB is the start element number for applying special boundary conditions

LE is the last element number for applying special boundary conditions

LBC is the boundary condition code to be applied to the elements defined by LB and LE above

Notes :

1 It is only necessary to use this keyword if the boundary condition codes differ from the default value of 1 and if the boundary condition codes have not already been set through file FILE.2

2 This keyword is used when boundary elements do not represent a free surface. For example, when boundary elements are used to model solid objects or the interface between different subregions, then these elements are identified by boundary condition codes different from the default value of 1. A simple example is that of a cube under uniaxial compression modelled with 6 elements. The bottom, back and left sides of the cube have constraints of movement, normal to each surface. (The bottom of the cube cannot move up or down in the Z direction and is given a boundary condition code of 5. Similarly, the back and left faces are given codes 3 and 4 respectively).

3 Valid boundary condition codes are as follows

Tractions specified (normal mined element)	1
Fixed (or rigid) element	2
Element cannot move in X-direction	3
Element cannot move in Y-direction	4
Element cannot move in Z-direction	5
The interface between two subregions	7

3 The end of special boundary condition input is indicated by one line of data containing three zeros

Keyword **END**

Purpose : Define an end of file marker

Format :

The keyword END is the only line required

Variables:

none

Notes :

1 This keyword can be used to define an end file marker even there is more data after this keyword. This can be useful sometimes when comparative runs are being carried out. It is not essential to use this keyword at all, but it is also to ensure that data format above the keyword is correct.

Keyword **HEADING**

Purpose : To define a user heading for a run

Format : USERHED

Variables:

USERHED Up to 64 characters

Notes :

1 The user is urged to vary this heading before every run. It is very two get two different runs confused if the user description does not differ from one run to the next

Keyword **INTEGR**

Purpose : To change integration accuracies to values different from the default values

Format : TR2,TR3,TR4,TR5,TR6,TR10
TRLB

Variables:

The above variables define the transition values at which different order Gauss quadrature formulae will be applied. The transition is based upon the separation ratio of the element or sub-element to the point in question. TRLB is a factor which is applied to TR2,TR3,...TR10 during the calculation of benchmarks. This is necessary because stress calculations require more Gauss quadrature points than do displacement calculations. A recommended value for TRLB is 0.7.

Notes :

1 It is usually not necessary to use the INTEGR keyword.

2 This keyword controls the accuracy with which the influence coefficient matrix is calculated. The variables TR2 to TR 10, together with SSW6 (see SSW keyword) control the integration accuracy. For example, if the integration accuracy is too low, then many elements (in module B503) will produce "TSELF ERRORS" greater than 5%. If this happens, then one might, typically, double all the values TR2 to TR10 and perhaps also SSW 6 to see whether the "TSELF ERRORS" are reduced. If only one or two elements produce high "TSELF ERRORS", then it may be better to check the geometry and shape of these elements and surrounding elements. Approximate guidelines for these variable (based on user experience) are given below.

Variable	Approximate Minimum	Maximum	Typical	Comment
TR2	3	300	6	About twice TR3
TR3	1.5	50	3	About twice TR4
TR4	1.0	25	1.8	About twice TR5
TR5	.5	10	1.0	About twice TR6
TR6	.2	3	0.5	About twice TR10
TR10	0	1	0.3	
SSW6	0.5	100	2	

Increasing the above values will improve the accuracy of calculation of influence coefficients in general. Note regarding variable TRLB: Stresses and displacements at benchmarks are also calculated using the above transition ratios (TR2 to TR10 and SSW 6). However, the stress coefficients are dependent upon an inverse distance cubed function. This means that more accurate integration is generally required. TRLB is a factor by which this integration can be improved (at the cost of extra computation time). Decreasing TRLB will increase the accuracy of calculation of benchmarks. Values between 0.4 and 1.0 are suggested, with a default of 0.7.

4 Two other variables printed in module B903 (Integration ratio for lump lumping and Integration ratio for element lumping) cannot be changed directly by the user, but are changed indirectly by the switch variable SSW 6.

Keyword **ISSW**

Purpose : To define integer control switches

Format :

ISW,IVAL
0,0

Variables:

ISW is the number of the switch to be changed
IVAL is the new value for that switch

Notes :

1 Use a line containing 0,0 to end input for this keyword

2 Twenty integer switches have been allocated for future use in the system. Most of these have not yet been used and are thus set to zero by default. Some of the swithces are for source code development purposes, and are thus of no direct concern to the everyday user. Switches which may be of use or interest to the user are as follows:

ISSW 1-4 Not used

ISSW 5 Produce debug information. This should not normally be used since it is primarily for program development and could substantially slow down the execution speed of programs.

ISSW 6 Loop Counter. The default is 5 meaning that screen messages for reading in nodes are displayed for every 5th node that is processed. Alternative values range between 1 and 50 and may be set by the user.

ISSW 7 Used internally. Do not use.

ISSW 8 Menu control 1 = on, 0 = off. If menu control has been set as off, then the user would not be asked for the name of each input datafile name. Instead, the default names, D201,D202 etc would be used.

ISSW 9-10 Used internally. Do not use.

ISSW 11 Iteration counter to keep track of the total number of iterations performed since the data base was initialized.

ISSW 12 Acceptable values are 2, 3 or 4. A value of 2 will produce less accurate results (usually) than a value of 4. Examples are given in Diering (1987) showing the effect of different levels of accuracy when evaluating element self effects.

ISSW 13 Printout verbosity. At present, 0 = full, whilst 1 will result in the suppression of page feeds, thereby producing a more compact form of printout.

ISSW 14	Switch 14 records the number of pages which have been printed (ie sent to the list device/channel). The page numbers are reset to zero when a new project is initialized (M101). thereafter it keeps incrementing by 1 until reset again using M101 or using the ISSW keyword.
ISSW 15	Switch 15 records the number of times that M501 has been run.
ISSW 16-19	Not used
ISSW 20	Number of decimal places for plotted numbers.

Keyword **ITERAT**

Purpose : Change default iteration parameters.

Format : NITER,OMEGA,DEL,NITSUB,ACCEL

Variables:

NITER	Maximum number of iterations permitted
OMEGA	Relaxation factor (applies to subiterations)
DEL	The required convergence ratio. This is the ratio of the change is displacement for the last and first iterations. Values of DEL range from 0.005 to 0.0005
NITSUB	Number of sub-iterations for each element.
ACCEL	Acceleration factor similar to OMEGA above, except it applies from one iteration to the next, instead of from one sub-iteration to the next as is the case for OMEGA

Notes :

1 It is normally NOT necessary to use this keyword

2 If convergence problems are encountered, then try reducing the value of OMEGA to between 0.4 and 0.8 whilst setting NITSUB = 1 and ACCEL = 1. Convergence should normally occur in from 10 to 30 iterations for single subregion problems and in from 30 to 100 iterations for multiple subregion problems.

Keyword **PRIM0**

Purpose : To define constant primitive stresses

Format : PXX,PXY,PXZ
PYX,PYY,PYZ
PZX,PZY,PZZ

Variables:

PXX Constant primitive stress in x-direction

PXY Constant primitive x-y shear stress = PYX

PXZ Constant primitive x-z shear stress = PZX

PYY Constant primitive stress in y-direction

PYZ Constant primitive y-z shear stress = PZY

PZZ Constant primitive stress in z-direction

Notes :

1 Positive stresses are compressive

2 The format may use three lines as shown above, or may be compressed into one line as

PXX,PXY,PXZ,PYX,PYY,PYZ,PZX,PZY,PZZ

3 Usually, the shear components are zero

4 It is imperative that the shear components are defined correctly if these are non-zero. It is imperative that the following equalities hold

PXY = PYX
PXZ = PZX
PYZ = PZY

5 These primitive stresses will automatically be applied to all elements whose boundary condition code (KOD) is equal to 1 denoting a mined element. They will not be applied to interface elements.

6 These stresses may be added to those defined by the PRIM1 keyword, by simply defining both keywords in the same data file

Keyword **PRIM1**

Purpose : To define depth dependent primitive stresses

Format : DXX,DXY,DXZ
DYX,DYY,DYZ
DZX,DZY,DZZ

Variables:

The variables DXX,...DZZ have units (stress/length) and are defined in similar way to PXX,....PZZ for constant primitive stresses defined for the PRIM0 keyword. The depth is defined here as being equal to the z-coordinate axis. For example, the actual primitive stress pxx in the x-direction is calculated from DXX as

$$pxx = DXX * z$$

where
z is the depth of the point in question

Notes :

1 Compressive stresses are positive

2 Often, the global z-axis for definition of nodal coordinates will be positive upwards, with the origin at surface. When this happens, all depths (z-coordinates) will be negative and DXX,...DZZ must also be negative to give compressive primitive stresses

3 The total primitive stresses are the sum of those defined by the keywords PRIM1 and PRIM0 if both have defined. ie:

$$pxx = DXX * z + PXX$$

4 Be sure to read the notes for the PRIM0 keyword

5 For example, if the horizontal stress in the X-direction is twice the vertical stress and the horizontal stress in the Y-direction is 1.5 times vertical, then for a vertical stress determined from overburden loading, with units of meters and the Z-axis pointing upwards with origin at earth's surface, typical values for PRIM1 would be :

-0.0540, 0.0000, 0.0000
0.0000,-0.0405, 0.0000
0.0000, 0.0000,-0.0270

Keyword **REPEAT**

Purpose : To define a number of repeatability images

Format :

```
NRPT
0,0,0                                          For object
XRPT,YRPT,ZRPT                                 For image 2

...
...
...

XRPT,YRPT,ZRPT                                 For image NRPT
KRPT(1),KRPT(2),KRPT(3), .... ,KRPT(NRPT)
```

Variables:

NRPT Number of images of repeatability to be used. The object is assumed here to be the same as image 1. The first true image is therefore treated here as image 2.

0,0,0 is the offset applied to the object. ie The object is an image which is not offset at all.

XRPT is the additive X-offset to be applied to an image
YRPT is the additive Y-offset to be applied to an image
ZRPT is the additive Z-offset to be applied to an image
KRPT has the value 0 or 1 for each image from 1 to NRPT. A value of 0 means that particular image is not used for the subregion. For each subregion there is one line of values KRPT (from 1 to NRPT)

Notes :

1 Up to 30 images may be used at present.

2 A fairly comprehensive description of the repeatability concept is given in Ref. 12. Before attempting a real run, any user would be advised to run the five parallel tunnels test example given in Ref. 12, and to try changing various parameters in those test files. A good check when using repeatability is to obtain a plot showing all the repeatability image. Note that abuse of the REPEAT keyword will yield very erroneous results.

Great care must be exercised to ensure that images do not overlap with each other or with the object. It is strongly recommended to plot the geometry with M301 to check that the definition of repeatability images is correct

3 As an example, consider a three subregion problem, in which repeatability applies only to the first subregion. Assume that there is one object and nine images, giving a total of 10 effective images. Therefore NRPT = 10 and the format for this keyword for this example would be:

REPEAT	Define the repeatability keyword
10	10 images (with object)
0,0,0	Zero offset for object = image 1
10,0,0	Offset for image 2
20,0,0	
30,0,0	
40,0,0	
-10,0,0	
-20,0,0	
-30,0,0	
-40,0,0	
-50,0,0	Offset for last image
1,1,1,1,1,1,1,1,1,1	All images for subregion 1
0,0,0,0,0,0,0,0,0,0	No images for subregion 2
0,0,0,0,0,0,0,0,0,0	No images for subregion 3

Keyword **SPTRAC**

Purpose : To apply special load conditions to elements over and above the primitive stresses

Format :

LB,LE
NB,NE,TX,TY,TZ

Variables:

LB and LE	are as defined above (or use 0,0) to end this section of input
NB	First function node to apply tractions
NE	Last function node to apply tractions
TX	Applied traction in X-direction
TY	Applied traction in Y-direction
TZ	Applied traction in Z-direction

Notes :

1 Examples of the use of this keyword are for applying loads to solid object problems where primitive stresses are inapplicable or for simulating the internal loads generated by caved or backfill material

2 It is also possible to apply separate tractions to each of the function nodes within a single element. This provides an approximate means of applying point loading

3 The line NB,NE,TX,TY,TZ is continuously read in until NE=9, whereupon the next line containing LB,LE is read.

For example

LB,LE	
1,3,TX,TY,TZ	Apply TX,TY,TZ to nodes 1 to 3
4,7,0,0,TZ	Apply 0 , 0,TZ to nodes 4 to 6
8,9,tx,ty,tz	Apply tx,ty,tz to nodes 8 to 9
LE,LE	Start with next group of elements

Keyword SSW

Purpose : To define real control switches

Format :

ISW,RVAL
0,0

Variables:

ISW is the number of the switch to be changed
RVAL is the new value (real) for that switch

Notes :

1 Use a line containing 0,0 to end input for this keyword

2 As with the ISSW keyword, only some of these are relevant to the user as follows:

SSW 2 Disk I/O monitor gives an indication of the amount of disk I/O to the random access files used by the program. Because this can be quite large, considerable time (and wear and tear) savings can be gained by putting the LUMSTOR.BE files into virtual disk. SSW2 is automatically set to zero for a new run.

SSW 3 Allowable curvature for elements. This value may be changed by the user, and will be used to trigger the warning message regarding curved elements. Possible values lie between 0.000 and 0.5. A value of 0.05 has proved adequate for almost all runs.

SSW 4 Allowable aspect ratio for elements. (Similar to SSW 3 above). This is the ratio of the longest to the shortest side of an element. Warning messages are issued if this ratio is exceeded.

SSW 5 This counter is for interest only, and gives an indication of the number of sample/integration points used to set up the influence coefficient matrix.

SSW 6 This is an important switch and can be used to change the amount of lumping that will be used in a run (and also the amount of disk space). Bigger value result in less lumping. Less lumping results in improved accuracy at the expense of increased disk space and execution time.

Typical values range between 0.8 and 4. Examples are given in Ref. 1. However, there is no hard and fast rule regarding the use of this lumping control. If its value is too low, and lumping is used extensively (producing many 1's in the lumping map of module (M502)) then results will be less accurate, and many "TSELF ERRORS" will be produced in module M503.

SSW 11, 12, 13 These are as set in line 2 of File 1.

SSW 14 Internal to program - do not use.

SSW 15 Internal to program - do not use.

SSW 16 Internal to program - do not use.

SSW 17 Internal to program - do not use.

SSW 18 Scale factor applied to displacement vectors before plotting

SSW 19 Scale factor applied to stress crosses before plotting (this is merely for compatibility with QUAD, since no modules of BEAP-DD will produce this type plot)

SSW 20 Scale factors for numbers before plotting. For example, if it is required to plot extension strains as micro-strains, instead of actual strain values, then a scale factor of 1.E6 or 1000000 would be used here.

Keyword **SUBS**

Purpose : **To define elastic properties, symmetry code and density for each subregion**

Format :

NSUB
MR,E,P,N,DENS

Variables:

NSUB number of subregions (Usually 1)

MR subregion number whose properties are being defined

E Young's modulus for the subregion

P Poisson's ratio for the subregion

N Symmetry code for the subregion

1	for no symmetry
2	for X symmetry
3	for Y symmetry
4	for z symmetry
5	for XY symmetry
6	for XZ symmetry
7	for YZ symmetry
8	for XYZ symmetry

Notes :

1 Most mining problems consist of one infinite subregion which contains all excavations to be modelled. If more subregions are to be modelled, then these would be defined under this keyword. Interface boundary elements would be defined in FILE.2. Note that the DENS variable should always be set equal to zero.

2 Default planes are defined at X=0, Y=0 and Z=0. Some or all of these planes can be activated using N as defined above. If different planes are required, then one should also use the SYM keyword

3 Nodes may be specified to lie on a plane of symmetry, but elements should never be defined on a plane of symmetry.

Keyword **SYM**

Purpose : To define planes of symmetry as being different from the default values 0,0,0.

Format :

XSYM,YSYM,ZSYM

Variables:

XSYM is the x-coordinate defining a plane of symmetry lying in the Y-Z plane at the specified X-coordinate

YSYM and ZSYM are defined as for XSYM

Notes :

1 The planes of symmetry are only activated if the relevant symmetry code is specified in the subregion definition section. Therefore, it is not sufficient to merely specify this Keyword if symmetry is required. If symmetry planes at X=0, Y=0 or Z=0 are being used, then it is not necessary to use this keyword

3.5 AN EXAMPLE OF THE FILE.3

An example of the FILE.3 data file for the cube excavation problem is given below

```
HEADING
CUBEX TEST
SUBS
1
1,100,.3,1,0
PRIM0
10,0,0,0,10,0,0,0,5
ITERATIONS
20,.9,.001,4,1.05
INTEGRATION
4,2,1.2,.8,.5,.4
0.50
ISSW
12,0
0,0
BENCH
1,11,5,0
Line 1
2,12,5,0
Line 1
3,13,5,0
Line 1
4,14,5,0
Line 1
5,15,5,0
Line 1
6,11,11,0
Diagonal
7,12,12,0
Diagonal
-99,0,0,0
END
```

4.0 DESCRIPTION OF PROGRAM MODULES

In the following paragraphs, each program module making up the BEAP system is described. Program modules may be run either via the program menu driver, or via DOS as a command. For example M204 is the fourth choice of menu number 2, or M204 may be run from DOS simply by entering:

M204

at the DOS prompt. It is worth noting here that DOS batch files can easily be built to enable many modules to be run at once. The user responses can also be built into response files which can be activated using the DOS piping capability. Refer to the DOS manual for further details.

When each program is run, the printout is written to a file TEMP.LST. If this printout is required as hard copy, then program control must be returned to DOS, and the file TEMP.LST sent to the printer using a DOS PRINT or COPY command. If programs are being run under batch control, then the PRINT commands can be built into the batch file. If the next program starts before the printing of the previous run is complete, then some of the printout can be lost. To avoid this situation, the DOS COPY command is sometimes preferable, since it waits until the printout is complete before proceeding.

BEAP Menu driver for BEAP and BEAP-M

This program enables BEAP to be run under menu control. Before this program will start, the files AUTOEXEC.BAT, CONFIG.SYS must have been previously adjusted as described under the installation procedures. In addition, there should be at least 400k free system memory available and at least 2 Megabytes of free disk space. In addition, the file MESAGE.BE must have been located in the subdirectory BEAP on the C drive (ie C:\BEAP\MESAGE.BE must exist).

BEMPOST Benchmark post-processor.

This program enables benchmark results to be prepared for plotting. It enables various failure criteria to be tested also, the results of which are contained in the printout.

Plot files which are generated are :

BEMPOST.1	User's first choice
BEMPOST.2	User's second choice (optional)
BEMPOST.D	Displacement vectors
BEMPOST.D3	Third component of displacement (ie the one not included in BEMPOST.D)

These files are in a binary format and require a separate plotting utility before they may be plotted. One example would be GEMCOM'S QUIKPLOT or GEMPLOT utilities. The format of all plot files is described separately, so that other plotting utilities may be modified to read these plot files.

BEMPOST enables coordinates and/or stresses to be transformed according to a user defined view point and origin. In addition, a box may defined to filter those benchmarks which are to be processed. This is useful, in that it enables benchmarks to be processed on a level by level or section by section basis.

BEMPOST will access the current database directly. In particular, it will read the benchmark results from file BENCH.BE. It will obtain subregion material properties from BOTH.BE.

Before BEMPOST may be run, the message file BEMPOST.MES must have located in the directory C:\BEAP (ie C:\BEAP\BEMPOST.MES).

M101 Initialize an entire new database (BEAP and BEAP-DD)

M101 will initialize all database files for new run. **NB** This includes the database files for BEAP-DD and BEAP-M if these programs are being run at the same time. It will create empty files (not necessarily with 0 bytes, however, since certain direct access files must contain an initial header record). A list of the files created is as follows:

List of files for BEAP, BEAP-M and BEAP-DD

Used by BEAP, BEAP-DD and BEAP-M :

BEMCO.	BE	Influence coefficients (BE)
BEMMESH.	BE	Plot file of BE mesh
BEMMESHE.	BE	Plot file of BE element numbers
BEMMESHN.	BE	Plot file of BE node locations (as X)
BEMPOL.	BE	Polygon master file
BENSTOR.	BE	Benchmark file (Binary)
BOTH.	BE	Basic information (internal to program)
BVECT.	BE	Right hand side vector
LIND.	BE	Lumping maps for (BE)
LUMPS.	BE	Geometric information for BE elements
LUMSTOR.	BE	Detailed information for BE elements
LUMSTOR2.	BE	Second file for BE elements
SELFCO.	BE	Element self-coefficients (BE)
VARCOM.	BE	Basic information (to supplement BOTH.BE)
XN.	BE	Nodal coordinates
TEMP.	LST	printout file for results

Only used by BEAP-M

COEF.	BD	Influence coefficients (BE-DD)
LIND.	BD	Lumping maps (BE-DD)
BVECT.	DB	Right hand side vector
COEF.	DB	Influence coefficients (BE)
LIND.	DB	Lumping maps (BE-DD)

Used by BEAP-DD (and hence BEAP-M also)

BEMCO.	DD	Influence coefficients (BE)
BEMMESH.	DD	Plot file for DD elements
BEMMESHE.	DD	Plot file of DD element numbers
BEMMESHN.	DD	Plot file of DD node locations (as X)
BEMSTEP.	DD	Store information for multi step runs
BVECT.	DD	Right hand side vector
LIND.	DD	Lumping maps (BE-DD)
LUMPS.	DD	Geometric information for DD lumps
LUMSTOR.	DD	Result information for DD elements
LUMSTOR2.	DD	More information for DD elements
LUMSTORG.	DD	Geometric information for DD elements
POLPLOT.	DD	Plot file for polygons
SELFCO.	DD	Self effects for DD elements
SHEET.	DD	File to store spreadsheet information
STORSTEP.	DD	Store information for multi step runs

M103 Initialize displacements and tractions

This is a useful module which initializes only the displacements and tractions for each element. It can be run if all geometric information has been read in correctly, but there was, for example, an error or adjustment in FILE.3, or if a new run is to be started with the same geometry, but only changes to the FILE.3 file. If the user is uncertain as to it's use, then it will be safer to run M101 and start a completely new run.

M201 Read in node and element geometry

This module reads the first of the user data input files (FILE.1). The user is prompted for the name of his file. Thereafter, various information is displayed to screen to show progress as the file is read. If a data input error is encountered, the program may crash. If this is the case, then correct the data error and run M101 and M201 again. If the data is read in successfully, then proceed with M202, M301 or M204 according to your requirements.

M202 Check nodes and elements (usually included in M203)

M202 will perform various checks on the element geometry. Aspect ratios, curvature and shape are checked. If unexpected values are encountered by the program, then a warning is displayed to the screen and to the print file TEMP.LST. The user should note these carefully and check the mesh to ensure that all messages are either corrected, or are explained and justified.

If corrections are required, then go back to M101 and start again. If numerous messages are displayed, then printout the file TEMP.LST to the printer to assist in the data file correction process.

M203 Read in element TYPE definitions and check

M203 will read in the second user file (FILE.2 as described earlier). This will alter the locations of the function nodes as well as initializing various boundary conditions and subregion interface codes. When it has been run, it will automatically perform the same set of checks as performed by M202 above. Therefore, if M203 is to be run, then it is not necessary to run M202 first. (ie the user will often run M201, followed by M203. This is acceptable.)

M204 Print selected elements or nodes (Detailed print)

This enables the user to obtain a printout of selected nodal coordinates (usually to facilitate checking the input data). In addition, locations and effective areas for the function nodes may be printed for each element as well as summary information for each element. The user may select which elements are to be printed, since a total printout could be 50 pages or more. Printout is directed to file TEMP.LST.

M205 Display selected elements to screen

This is similar to M204, except that information is directed to screen, enabling the user to 'browse' through the data before obtaining a hard copy with M204.

M209 Print all element geometry (batch mode of M204)

This is also similar to M204, except that all elements are automatically printed, without any prompting from the user. This is useful in BATCH runs in which user interaction is required to be minimized.

M301 Prepare a 3-D view of mesh/elements/nodes

This is an extremely useful debugging tool. The user can obtain a 3-D isometric view, and X-Y plan plot, or X-Z, Y-Z section plots (ie looking north, south, east or west). The user may also select whether only those elements which are pointing towards or away from the view point may be plotted. No hidden line removal is performed, but usually by choosing an appropriate view point, a reasonable image may be obtained.

The user may also elect to prepare only selected elements for plotting. Plot information is routed to the files BEMMESH.BE, BEMMESHE.BE and BEMMESHN.BE.

M401 Read in other input data

This module will read in the remainder of the users input from the file (FILE.3). Each keyword is decoded and relevant information displayed to the screen. If a data error is encountered here, then the data file should be corrected. Then run M103 (or M101 if necessary followed by M201 etc) and then run M401 again. Usually there will not be any problems running this module several times, (for example if there is an error with the benchmark data). However, the user should be wary of files which contain the SPTRAC keyword since this is an additive process. (ie Running the data file twice will add twice the specified tractions to specified elements)

This module is also used to set the default plot file preparation parameters (ie stress/vector scale factors, number of decimal places, scale factor for numbers written out and character sizes). See the SSW keyword for further detail.

M402 Print header page and selected information

This module provides a one page printout (to TEMP.LST) showing relevant general information, such as header description, primitive stresses, numbers of benchmarks, elements and nodes etc. Information is written to TEMP.LST.

M501 Apply tractions due to primitive stresses

M501 will apply the primitive tractions (as defined by the PRIM0 and PRIM1 keywords to all elements whose boundary condition is 1 representative of a 'mined' surface. The elements which are affected are displayed to the screen. If there is uncertainty as to what values have been applied, then the user can run M204 or M205 to check the actual traction values at element function nodes. If the user attempts to run this module twice, a warning message is issued. This is because the primitive tractions will be added to whichever tractions already exist at each element. (ie total tractions = primitive tractions + tractions from SPTRAC keyword).

M502 Calculate lumping maps (BE-BE)

This module calculates the lumping maps for boundary elements (BE-BE). The effect of each element on each other element is assessed and the type of lumping or element-element interaction required is assessed. The following codes are displayed to the screen and to the print file TEMP.LST.

	Description	Size (bytes)
1	Lump-lump coefficients (for remote elements)	36
2	Lump-lump coefficients (partial)	144
3	Lump-element coefficients	324
4	Element-lump coefficients	324
5	Element-lump coefficients(partial)	1296
6	not used	-
7	Element-element interaction (elements are close to each other)	2916
8	Adjoining element (connected at one or more nodes)	2916
9	Self effect (receiver element = emitter element)	2916

Interface elements use twice the space required by other elements.

The total size which will be required by the files BEMCO.BE and SELFCO.BE by M503 is calculated and reported in TEMP.LST. The user should inspect the printout and check whether sufficient disk space is available for the remainder of the run. If the space required by these files is not available, then the amount of space may be reduced (at the expense of reduced accuracy) by decreasing the value of real switch number 6

```
SSW
6,1.6
0,0
```

For example, the above segment of data file FILE.3, if read in via M401 will set the lumping transition ratio to 1.6. Values between 0.8 and 4 will usually be okay. If too much lumping is being used, numerous TSELF errors will be printed when M503 is run. The correct choice for this ratio is best found by experience and the accuracy requirements of the user.

The lower the value of this ratio, the more 1's will be produced in the lumping map and the less will be the storage requirements. (see reference 9 for further information)

M503 Calculate influence coefficients (BE-BE)

This module calculates the effect or influence of each function node on every other function node. The level of detail or accuracy which is used for each set of coefficients depends upon the integration parameters as set by the INTEGR keyword. If this keyword is not specified, then the default values will be used, which should be acceptable for most runs. See further details under the INTEGR keyword description.

Various information is displayed to both screen and print file (TEMP.LST) as this module progresses. Usually, two lines are printed for each element, once it has been processed (as a receiver element). These lines are as follows:-

Element No., Maximum etc

The files BEMCO.BE, SELFCO.BE and BVECT.BE are updated by M503.

Various (sometimes kryptic) information is written to the printer by this module. Much of this information was used when developing the program and is not usually relevant to the everyday user. Usually, two lines of printout are produced for each lump (except when planes of symmetry have been defined). These lines are as follows:

Element No., Maximum Element No., Current Points, Total Points, Element No., Max % Error in T-SELF coefficients = - - - .

Current points is the number of integration points which were used to evaluate the influence coefficients for that element. Typical values range from 10,000 to 25,000. (Higher values generally reflect increased integration accuracy).

Total points is a cumulative counter (see SSW 5) reflecting the total computation effort involved in evaluating the co-efficient matrix.

The % error in TSELF reflects the accuracy with which co-efficients for a particular element have been calculated. The following comments might assist the user in assessing these TSELF error values.

Range of Errors	Frequency	Action
0-1%	Most or all elements	Decrease integration accuracy using INTEGR keyword. Excess time is being used to calculate the coefficient matrix.
1-2%	Most or all elements	Ideal situation - do nothing
2-5%	Some elements	Do nothing unless special accuracy is required close to these elements.
2-5%	Most or all elements	Suggest an increase in integration accuracy.
5-15%	Some elements	Check these elements for geometry/shape. Not too important if benchmarks or stresses are not required close to these elements.
5-15%	Most or all elements	Increase integration accuracy. Check that all elements have been correctly defined.
>15%	Some elements	Try to fix the shape of these elements unless they are triangular. they are triangular, try to renumber, so that element degeneracy is at corner with largest internal angle. See Note 1 below.
>15%	Most or all elements	Increase integration accuracy. Check for mesh errors. Entire problem may be ill defined for a three dimensional boundary element analysis. (The exception occurs when a semi infinite problem is being modelled).

Note 1:

Incomplete subregions are those where elements are terminated abruptly far from the region of interest. (Examples are the earth's surface or the end of a long tunnel). The incomplete subregion will produce locally large TSELF errors near the area of termination which are not serious if remote from areas where results are required.

M504 Solve equations (BE)

The boundary element equations are solved in an iterative manner. If the problem has been correctly defined, then the displacements and tractions should slowly converge towards the 'correct' solution. The rate of convergence is controlled by the ITERAT keyword. If this keyword is not present in the FILE.3 data file, then the default values will be used. If the error (actually an estimate of the error based on the change of displacement from one iteration to the next) displayed at each iteration is not converging satisfactorily, then the element mesh or configuration may be in error. If a multiple subregion problem is being modelled, then different values are required for the ITERAT keyword. See the separate section on subregion modelling.

This module will often take several hours to run, and is thus usually best run together with M503 and M507 in batch mode.

M504 will read the array coefficients and right-hand side vector of the equations from the files BEMCO.BE, SELFCO.BE and BVECT.BE. The updated displacements and tractions are written into files LUMSTOR.BE and LUMSTOR2.BE. In addition, other information is updated in files BOTH.BE and LUMPS.BE.

M507 Calculate benchmarks (BE)

M507 will evaluate the stresses and displacements at each specified benchmark point (as set using the BENCH keyword). Results are written to the printout file TEMP.LST and are also written the file BENCH.BE (binary format) for subsequent processing by BEMPOST or M603.

It is suggested that benchmarks do not be located closer to an element than one fifth the length of the longest side of the element. This rule may be 'bent' sometimes, but only by sufficiently experienced users. If benchmarks are too close to an element, then this is usually characterized by artificially high tensile stresses being calculated.

The accuracy for calculations for benchmarks which are close to an element are particularly dependent upon the specified integration accuracy (see INTEGR keyword).

Benchmark printouts contain four main sections for each benchmark point. These are:

- Location and displacements
- Stresses
- Principal Stresses
- Accuracy/Integration information

Each section is described below.

(i) Location:

IBEN	IDB	NAMEB
XB	YB	ZB
UB	VB	WB

where	IBEN	Benchmark number
	IDB	User number for benchmark
	NAMEB	User description for benchmark
	XB,YB,ZB	Location of benchmark
	UB,VB,WB	Displacement components of benchmark

(ii) Stresses:

IXX	IXY	IXZ
	IYY	IYZ
TXX		IZZ
TXY	TYY	
TXZ	TYZ	TZZ

Where IXX ... IZZ are induced stresses. For example, IYY is the induced normal stress in the Y direction, while IXZ is the induced shear stress components in the XZ plane. Remember Total Stresses = Induced stresses + Primitive stresses where primitive stresses are as defined by the PRIM0 and PRIM1 keywords.

TXX, ... TZZ are total stress components similar to IXX ... IZZ above.

(iii) Principal Stresses

PMIN	PINT	PMAX
DMIN	DINT	DMAX
DDMIN	DDINT	DDMAX

Where:

PMIN, PINT, PMAX are minimum, intermediate and maximum principal stresses

DMIN, DINT, DMAX are dip vectors for the orientation of the principal stresses measured from the X-Y plane towards the -ve Z axis (i.e. a lower hemisphere projection).

DDMIN, DDINT, DDMAX Defines the orientation of the principal stresses in the X-Y plane. A value of zero implies that the projection of the principal stress onto the X-Y plane lies along the +ve X-axis. 90 corresponds with the +ve Y-axis, + or - 180 corresponds with the -ve x axis and so on.

(iv) Accuracy/Integration Information

Nearest	at XL, YL
NPTS	MOST =
NUMLUM	NUMELT =
CLOSEST	RATIO =
SUB-REGION	

where Nearest	=	Element closest to the benchmark
XL	=	Local X coordinate of element point closest to benchmark
YL	=	local Y coordinate of element point closest to benchmark
NPTS	=	Number of integration points which were used to evaluate the benchmark
MOST	=	Highest number of integration points used for any one element to evaluate the benchmark.
NUMLUM	=	Number of elements which were not divided into subelements.
NUMELT	=	Number of sub-elements used
CLOSEST	=	Closest distance encountered between elements (Boundary surface) and the benchmark
RATIO	=	Minimum ratio encountered for the ratio of distance squared divided by the area weight of an integration point (See notes below).
SUBREGION	=	Subregion number which contains the benchmark.

Close scrutiny of the above information is usually not necessary for each benchmark. However, it can be useful in deciding whether a particular benchmark is likely to be reliable or not. Benchmark values will tend to be unreliable if (1) the points are too close to the boundary element surface or (ii) if the integration accuracy is too low or (iii) if the points are too close to the edge of an element. The usual symptoms of the above three cases are as follows:

(i) Ratio has a value of less than 9. The minimum principal stress often shows an unrealistically high tensile stress.

(ii) RATIO has a value of less than 9, and MOST has a value of 36 or even 9 or 4. This indicates that although the benchmark was close to an element, automatic refinement of the integration did not take place.

(iii) If CLOSEST is less than one fifth of the square root of the area of the element closest to the benchmark, and XL and/or YL are close to + or - 1, then it is likely that that benchmark is too close to the edge of an element. Other symptoms are unexpectedly large shear stresses or principal stress orientations which change very rapidly from one point to the next.

Thus the information provided in the fourth section of the benchmark printout can assist the user to identify unreliable benchmark points. The user should of course exercise caution when defining the benchmark locations so as to avoid the above uncertainties.

M508 Calculate surface stresses and normal displacements

M508 will evaluate the surface stresses within an element. They are printed in a local coordinate system which is intrinsic to each element, and depends upon the node numbering for that element. Displacements in this local coordinate system are also printed.

Often, the normal displacement (Z-component as printed) is useful for assessing displacements normal to the excavation surface. Similarly, the stresses at the centre of the element are also useful. The stresses within an element are constrained to vary in a linear manner across the surface of the element. Therefore, in areas of high stress gradient, the surface stresses as calculated and displayed will not always be as accurate as one would hope. In other words, great care should be exercised, or a finer mesh should be used to better define/model the high stress gradient.

This module will also enable various printout options. The user may select from 1 to 9 surface stress points per element. Recommended from and to node selections are 1 to 9, 1 to 4, or 9 to 9. In addition, improved paging has been introduced. Finally, the user can also suppress the lines containing the coordinate transformation and primitive stress information.

Surface stresses may be calculated at selected elements and at selected locations within each element. At present, nine locations are reported automatically at each selected element. The ninth location is always at the centre of the element and will usually be the most reliable location for surface stresses. The others are located somewhere between the centre of the element (at a local coordinate of 0) or the edge of the element (local coordinate of 1). The recommended value for this local coordinate is 0.6667. The information reported for each element is as follows:

(i) Element identification (1 line)

(ii) Point number, coordinates, displacements, stresses (9 lines). Displacements and stresses are in the local coordinate system intrinsic to each element. This local coordinate system depends upon the manner in which the nodes for that element have been numbered by the user. The local X-axis increases approximately in the direction of the first to the second nodes of the element. The local Y-axis increases approximately in the direction of the second to the third nodes of the element, while the local Z-axis is always outwards normal to the element. Displacements are reported as local x, local y, local z. Stresses are reported as xx, xy, xz, yy, yz, zz (also in local coordinates).

(iii) Primitive stresses at the centre of the element. The format is similar to (ii) above.

(iv) Direction cosines of the local coordinate system with respect to the global coordinate system.

M509 Convert surface stresses into benchmark values

This module is used to convert surface stress results at element centres into benchmarks. This will enable the surface stresses at element centres to be processed using BEMPOST. The total number of benchmarks is automatically updated to reflect the new surface stress results. It should be noted that at interface elements, results are only produced for the softer material. Probably the most useful aspect of this module is to enable principal stress plots to be produced via BEMPOST for excavation surfaces at irregular orientations.

M601 Print selected element results

This module is basically the same as M204, except that it does not print out geometric nodal coordinates. It is used here to enable the user to print out the tractions and displacements for each (or selected) element(s). Graphical processing of tractions and displacements is also available through M701.

M602 Print all element results (batch mode of M601)

This module is the same as M601, except that it will print out results for all elements in batch mode. It will often be used with M503, M504, M507, M508 as part of an overnight run.

M603 Print selected benchmark results

This module enables selected benchmark results (including the surface stress results mentioned in M509) to be printed. The format of the results is the same as that produced by module M507. The program is run interactively, with the user selecting the required benchmarks.

M701 Prepare selected element results for plotting

This module enables a plot file (BEMMESH.BE) to be prepared for plotting or graphic display. Tractions or Displacements at each function node for selected elements are prepared as numbers.

The user can obtain a 3-D isometric view, and X-Y plan plot, or X-Z, Y-Z section plots (ie looking north, south, east or west). This plotfile may be plotted together with a mesh file as prepared by M301. If the plotfile BEMMESH.BE from a previous run is needed, then the file BEMMESH.BE should be copied to some other filename before this module is run, since any previous information in the file BEMMESH.BE will be overwritten.

M901 Print nodes and element in data file format

The user can use M901 to obtain a listing (formatted) of the global nodes and element/node definitions. This is often useful for debugging a mesh preparation error. For example, if it is known that there are errors in elements 34 and 45, which are connected to nodes 12,15,23,56,78,34 and 108, then these selected elements and nodes can be conveniently printed onto a single page for detailed checking.

M902 As for M901, with reversed element ordering

This module enables the user to obtain a printout/data file in which the node numbers for each element have been reversed. This is useful when the user has spent many hours numbers all nodes

back to front! However, if you use it, then extra care should exercised to check that final nodes for each element are correct. At present it has only been tested for elements containing 3 or 4 nodes. It will not work for elements containing 8 or 9 nodes (ie curved elements).

M903 Display/modify number of nodes, benchmarks and elements

This module enables the user to adjust the numbers of elements, nodes and benchmarks. It is particularly useful, when one wishes to remove some benchmarks out of the database.

M905 Load in a new message file

The message file (MESAGE.BE) is a direct access file which is used by the menu drivers and many modules and it contains the menu and many prompt descriptions. This same prompts are contained in an ASCII file (MESSB) which the user can edit using standard text editors. Provided that care is exercised to ensure that line numbers for each message are not altered, then the user may edit the ASCII message file and then update the 'system' message file in subdirectory C:\BEAPM\ using M905.

In this manner, much of the program could be translated into French for example. Alternatively, any prompt which the user finds misleading can be modified.

5.0 RUNNING BEAP

People using BEAP for the first time will run modules via the menu driver. However, as more familiarity is gained, it usually becomes easier to run modules directly from DOS or as part of batch files. The following is an example of a run scenario for a typical underground problem.

Draw up mesh on paper

List node coordinates and element/node definitions on paper

Capture node and element info into **FILE.1** using a text editor. Although the filename is referred to as FILE.1 here, remember that any file name may be used although the .1 extension is recommended. Be sure to make full use of the 16 character description allowed for each element. This is VERY useful for later on in the program. The small extra effort here to carefully name elements is worth it!

Create files **FILE.2** and **FILE.3** in a similar way. Often these will be created by copying these files from a previous run, and merely editing the relevant changes for the new run.

M101 Create a new database

M201 Read in FILE.1

Often there will be a data error here for a first time run. In this case, correct the data file and try again (ie M101 and M201)

If M201 has completed successfully, then run **M203** to read FILE.2 and carry out basic checks. Closely observe the displayed messages. If necessary, printout the file TEMP.LST to obtain a hard copy of these warnings. M204 may also be used to print out selected problem elements. Make corrections to the input data files and try again until all warnings have been eliminated or reasonably explained.

M301 Run this module now to prepare a plot file for the mesh. If a selected area is known to be in error, then select only those elements in the problem area. Make any required corrections and go back to running M101 again. It may also be necessary to run M401 here with a small data file containing the SSW and ISSW keywords to set the character size for the plots.

Use a plotter/display program to **view the plotfile** on the screen or plotter. (See separate section on plotfiles).

Once the data files FILE.1 and FILE.2 have been checked, then use **M204** to obtain a hard copy of the input element geometry.

M401 Read in the remaining input data including benchmark locations.

M402 Print out the Header page for this run.

M501 Assign the primitive tractions for each element.

M502 Calculate lumping maps. Copy the print file TEMP.LST to the printer and examine whether sufficient disk space is available to store the arrays BEMCO.BE and SELFCO.BE which will be created when M503 is run.

Then use a text editor to create a batch file (eg RUN.BAT) which will run all the number crunching modules at once). Typically, this would be as follows

```
M503            Calculate influence coefficients
COPY temp.lst lpt1   (copy print file to printer)
M504            Solve equations
COPY temp.lst lpt1
M507            Calculate benchmarks
COPY temp.lst lpt1
M508            Calculate
COPY temp.lst lpt1
M602            Print results at all elements.
COPY temp.lst lpt1
```

(The next day!) **examine the printout** to see whether the errors listed under M503 are acceptable, whether the equation solution process (M504) has converged, and whether the remainder of printout from M507, M508 and M602 appears reasonable. Do not take it for granted that if the program has run and printed results that everything is correct. Spend a few hours looking for strange results or unexpected results. For example, check that bottoms of excavations are tending to move upwards, and tops down. Check a few benchmarks at remote locations to see whether these are close to the applied primitive stresses. Check the benchmark printout to see that no benchmarks are closer to an element than was expected. (the closest separation between each benchmark and the closest element is printed). The more you check the results, the better will be your understanding of what is happening in the model.

COPY BEMMESH.BE MESH.BE (take a backup copy of the plot file which contains your best mesh plot to date).

It may be necessary to run M401 here to adjust plot parameters such as number of decimals and character size. (Use keywords ISSW and SSW here).

M701 prepare a plotfile of selected element results.

M509 Calculate stresses at each element centre so that these can be treated as benchmark along with those calculated using M507.

BEMPOST Perform various runs to check benchmark values for failure subject to a selected criterion or to prepare plot files.

6.0 REFERENCES

1 Deist, F H and Georgiadis, E, A computer system for three-dimensional elastic analysis using a boundary element approach. Chamber of Mines of South Africa, Research Report No 43/76, Project No. GS1510 (1976).

2 Hocking, G, Development and Application of Boundary Integral and Rigid Block Methods for Geotechnics. PhD Thesis, University of London, UK (1977).

3 Watson, J O and Cowling, R, Application of three-dimensional boundary element method to modelling of large mining excavations at depth. 5th Int Conf on Num Meth in Geomechanics, Nagoya (1985).

4 Diering, J A C, Further Developments of the Boundary Element Method with Applications in Mining. MSc Thesis, University of the Witwatersrand (1982).

5 Lachat, J C, A further Development of the Boundary Integral Technique for Elastostatics. PhD Thesis, University of Southampton, UK (1975).

6 Lachat, J C and Watson, J O, Effective Numerical Treatment of Boundary Integral Equations : A formulation for three-dimensional elastostatics. Int Jnl for Num Meth in Eng, 10, (1976) 991 - 1005.

7 Bannerjee, P K and Butterfield, R, Boundary Element Methods in Engineering Science. McGraw Hill (1982).

8 Watson, J O, Hermitian Cubic boundary elements for plane problems of fracture mechanics. Int Jnl of Struct Mech and Materials Sc. Ed J Gittus (1982).

9 Diering, J A C, Advanced elastic analysis of complex mine excavations using three-dimensional boundary element techniques. PhD Thesis Pretoria University, South Africa (1987).

10 Cruse, T A, Numerical solutions in three-dimensional elastostatics. Int Jnl Solids and Structures, 5, (1969) 1259 - 1274.

11 Brebbia, C A. The Boundary Element Method for Engineers. Pentech Press (1980).

12 Diering, J A C and Stacey, T R. Three-dimensional stress analysis : A practical planning tool for mining problems. Proc 20th Int Symp APCOM, Johannesburg, SAIMM, 1, (1987) 33 - 42.

13 Hoek, E and Brown, E T, Underground excavations in rock, IMM, London (1982).

14 Diering, J A C and Yu, Y S. Three-dimensional stress analysis : A formulation for mining problems. Computers and Geotechnics - Special issue on Mining Vol 5 No 2 Elsevier Appl Sc Publ (1988)

APPENDIX 1

EXAMPLE DATA FILES

The following pages contain test data files for several BEAP runs which demonstrate various aspects of the program.

Test data file : BOX.1

```
9,6
0,0,0
1,0,0,0
2,1,0,0
3,1,1,0
4,0,1,0
5,0,0,1
6,1,0,1
7,1,1,1
8,0,1,1
9,1.0,.5,.5
1,4,1,4,3,2
BOTTOM
2,4,7,6,2,3,0,0,0,0,9
FRONT   NOTE THAT 0 NODES DOES NOT WORK PROPERLY YET
3,4,4,8,7,3
RIGHT
4,4,8,4,1,5
BACK
5,4,6,5,1,2
LEFT
6,4,8,5,6,7
TOP
```

Test data file : BOX.2

```
1,6,1,1,1,1,0, 2,0,0
0,0,0,0,0,0,0,0,0,0
```

Test data file : BOX.3

```
HEADING
Cube test run
SPTRAC
6,6
1,9,0,0,-10
0,0
SUBS
1
1,10,.3,1,0.000
BOUNDR
1,1,5
2,2,1
3,3,1
4,4,3
5,5,4
0,0,0
ITERATIONS
50,0.8,.0005,3,1.02
INTEGRATION
1.5,1.0,.5,.4,.3,0
.70
ISSW
11,0
12,3
0,0
SSW
6,100.0
0,0
BENCH
1,.5,.5,.5
 AT CENTRE
2,.4,.4,.4
AT .4
3,.3,.3,.3
AT .3
4,.5,.5,.2
AT .2
5,.5,.5,.8
AT .8
-99,0,0,0
END
```

Test data file : SPHERE.1

```
19,3
0,0,0
 1 .000000 .000000 1.00000 .000000 90.0000
 2 .707107 .000000 .707107 .000000 45.0000
 3 .612373 .612373 .500000 45.0000 30.0000
 4 .000000 .707107 .707107 90.0000 45.0000
 5 1.00000 .000000 .000000 .000000 .000000
 6 .707107 .707107 .000000 45.0000 .000000
 7 .000000 1.00000 .000000 90.0000 .000000
 8 .382684 .000000 .923879 .000000 67.5000
 9 .353554 .353554 .866025 45.0000 60.0000
 10 .000000 .382684 .923879 90.0000 67.5000
 11 .732963 .303603 .608761 22.5000 37.5000
 12 .303604 .732963 .608761 67.5000 37.5000
 13 .923880 .000000 .382683 .000000 22.5000
 14 .874849 .362374 .321439 22.5000 18.7500
 15 .683013 .683013 .258819 45.0000 15.0000
 16 .362375 .874849 .321439 67.5000 18.7500
 17 .000000 .923880 .382683 90.0000 22.5000
 18 .923880 .382683 .000000 22.5000 .000000
 19 .382684 .923879 .000000 67.5000 .000000
1,9,4,3,2,1,12,11,8,10,9
TOP
2,9,3,6,5,2,15,18,13,11,14
FRONT
3,9,4,7,6,3,17,19,15,12,16
SIDE
```

Test data file : SPHERE.2

```
1,3,1,1,1,1,0,2,0,0
0,0,0,0,0,0,0,0,0,0
```

Test data file : SPHERE.3

```
HEADING
SPHERE WITH 3 ELTS AND POISSONS RATIO = 0.3 (WITH FIXED COORDS)
SUBS
1
1,1,.30,8,0
PRIM0
0,0,0,0,0,0,0,0,1
INTEGRATION
4,2,1.4,.9,.6,.4
0.50
BENCH
1,1.05,0,0
X-AXIS
2,1.10,0,0
X-AXIS
3,1.15,0,0
X-AXIS
4,1.20,0,0
X-AXIS
5,1.30,0,0
X-AXIS
6,1.40,0,0
X-AXIS
7,1.50,0,0
X-AXIS
8,1.75,0,0
X-AXIS
9,2.00,0,0
X-AXIS
10,4,0,0
X-AXIS
11,0,0,1.05
Z-AXIS
12,0,0,1.10
Z-AXIS
13,0,0,1.15
Z-AXIS
14,0,0,1.20
Z-AXIS
15,0,0,1.30
Z-AXIS
16,0,0,1.40
Z-AXIS
17,0,0,1.50
Z-AXIS
18,0,0,1.75
Z-AXIS
19,0,0,2.00
Z-AXIS
20,0,0,4.00
```

Z-AXIS
21,0.7424621,0.7424621,0
45 DEGREE
22,0.7778175,0.7778175,0
45 DEGREE
23,0.8131728,0.8131728,0
45 DEGREE
24,0.8485281,0.8485281,0
45 DEGREE
25,0.9192388,0.9192388,0
45 DEGREE
26,0.9899495,0.9899495,0
45 DEGREE
27,1.0606602,1.0606602,0
45 DEGREE
28,1.2374369,1.2374369,0
45 DEGREE
29,1.4142136,1.4142136,0
45 DEGREE
30,2.8284271,2.8284271,0
45 DEGREE

Test data file : CUBEX.1

```
8,6
0,0,0
1,0,0,-5
2,10,0,-5
3,10,10,-5
4,0,10,-5
5,0,0,5
6,10,0,5
7,10,10,5
8,0,10,5
1,4,1,2,3,4
BOTTOM
2,4,3,2,6,7
FRONT
3,4,3,7,8,4
RIGHT
4,4,5,1,4,8
BACK
5,4,2,1,5,6
LEFT
6,4,7,6,5,8
TOP
```

Test data file : CUBEX.2

```
1,6,1,1,1,1,0,2,0,0
0,0,0,0,0,0,0,0,0,0
```

Test data file : CUBEX.3

```
HEADING
CUBEX TEST
SUBS
1
1,100,.3,1,0
PRIM0
10,0,0,0,10,0,0,0,5
ITERATIONS
20,.9,.001,4,1.05
INTEGRATION
4,2,1.2,.8,.5,.4
0.50
ISSW
12,0
0,0
BENCH
1,11,5,0
BENCH 1
2,12,5,0
BENCH 2
3,13,5,0
BENCH 3
4,14,5,0
BENCH 4
5,15,5,0
BENCH 5
6,11,11,0
BENCH 6
7,12,12,0
BENCH 7
8,10.5,5,0
BENCH 8
9,5,10.5,0
BENCH 9
10,5,5,5.5
BENCH 10
11,5,5,5.6
BENCH 11
12,5,5,5.7
BENCH 12
13,11,11,1
BENCH 13
14,11,11,2
BENCH 14
15,11,11,3
BENCH 15
16,11,11,4
BENCH 16
17,11,11,5
BENCH 17
```

18,11,11,6
BENCH 18
19,11,11,7
BENCH 19
20,11,11,8
BENCH 20
-99,0,0,0
END

Test data file : CUBEX.HED

HEADING
CUBEX TEST TO CHANGE THE HEADING
END

Test data file : CUBEX.BEN

BENCH
1,11,5,0
BENCH 1
2,12,5,0
BENCH 2
3,13,5,0
BENCH 3
4,14,5,0
BENCH 4
5,15,5,0
BENCH 5
6,11,11,0
BENCH 6
7,12,12,0
BENCH 7
8,10.5,5,0
BENCH 8
9,5,10.5,0
BENCH 9
10,5,5,5.5
BENCH 10
11,5,5,5.6
BENCH 11
12,5,5,5.7
BENCH 12
-99,0,0,0
END

Test data file : FIVE.1

```
58,46
1800,4800,2600,OFFSETS TO COORDINATES
  1 1762 4746 2646
  2 1762 4785 2635
  3 1762 4746 2566
  4 1764 4785 2566
  5 1725 4746 2527
  6 1738 4785 2540
  7 1859 4746 2643
  8 1859 4785 2646
  9 1803 4746 2589
 10 1805 4785 2590
 11 1841 4746 2529
 12 1845 4785 2528
 13 1975 4746 2627
 14 1939 4785 2626
 15 1679 4746 2647
 16 1677 4785 2645
 17 1679 4746 2566
 18 1677 4785 2564
 19 1708 4746 2527
 20 1719 4785 2540
 21 1846 4746 2642
 22 1839 4785 2646
 23 1858 4746 2549
 24 1858 4785 2549
 25 1802 4746 2527
 26 1802 4785 2527
 27 1925 4746 2627
 28 1975 4746 2603
 29 1967 4785 2600
 30 1932 4746 2550
 31 1948 4785 2572
 32 1764 4824 2655
 33 1764 4862 2655
 34 1765 4824 2554
 35 1764 4862 2554
 36 1862 4824 2700
 37 1863 4862 2721
 38 1863 4824 2642
 39 1863 4862 2642
 40 1839 4862 2642
 41 1864 4824 2552
 42 1863 4862 2553
 43 1917 4785 2626
 44 1882 4746 2592
 45 1916 4785 2600
 46 1893 4746 2548
 47 1918 4785 2572
 48 1716 4824 2635
```

```
 49 1716 4862 2653
 50 1717 4824 2553
 51 1718 4862 2585
 52 1850 4824 2700
 53 1851 4862 2722
 54 1839 4824 2642
 55 1815 4824 2642
 56 1815 4862 2642
 57 1816 4824 2552
 58 1814 4862 2553
  1 4  15  16  2  1
FIVE EX.46
  2 4  3  1  2  4
FIVE EX.46
  3 4  5  3  4  6
FIVE EX.46
  4 4  18  16  15  17
FIVE EX.46
  5 4  20  18  17  19
FIVE EX.46
  6 4  20  19  5  6
FIVE EX.46
  7 4  17  15  1  3
FIVE EX.46
  8 4  19  17  3  5
FIVE EX.46
  9 4  4  2  16  18
FIVE EX.46
  10 4  6  4  18  20
FIVE EX.46
  11 4  21  22  8  7
FIVE EX.46
  12 4  23  7  8  24
FIVE EX.46
  13 4  11  23  24  12
FIVE EX.46
  14 4  10  22  21  9
FIVE EX.46
  15 4  26  10  9  25
FIVE EX.46
  16 4  26  25  11  12
FIVE EX.46
  17 4  9  21  7  23
FIVE EX.46
  18 4  25  9  23  11
FIVE EX.46
  19 4  24  8  22  10
FIVE EX.46
  20 4  12  24  10  26
FIVE EX.46
  21 4  27  43  14  13
FIVE EX.46
  22 4  28  13  14  29
FIVE EX.46
```

```
  23 4  30  28  29  31
FIVE EX.46
  24 4  45  43  27  44
FIVE EX.46
  25 4  47  45  44  46
FIVE EX.46
  26 4  47  46  30  31
FIVE EX.46
  27 4  44  27  13  28
FIVE EX.46
  28 4  46  44  28  30
FIVE EX.46
  29 4  29  14  43  45
FIVE EX.46
  30 4  31  29  45  47
FIVE EX.46
  31 4  48  49  33  32
FIVE EX.46
  32 4  34  32  33  35
FIVE EX.46
  33 4  51  49  48  50
FIVE EX.46
  34 4  51  50  34  35
FIVE EX.46
  35 4  50  48  32  34
FIVE EX.46
  36 4  35  33  49  51
FIVE EX.46
  37 4  52  53  37  36
FIVE EX.46
  38 4  38  36  37  39
FIVE EX.46
  39 4  41  38  39  42
FIVE EX.46
  40 4  56  53  52  55
FIVE EX.46
  41 4  58  56  55  57
FIVE EX.46
  42 4  58  57  41  42
FIVE EX.46
  43 4  55  52  36  38
FIVE EX.46
  44 4  57  55  38  41
FIVE EX.46
  45 4  39  37  53  56
FIVE EX.46
  46 4  42  39  56  58
FIVE EX.46
```

Test data file : FIVE.2

```
1,46,1,1,1,1,0,2,0,0
0,0,0,0,0,0,0,0,0,0
```

Test data file : FIVE.3

```
HEADING
FIVE EXCAVATIONS
ITERATIONS
20,0.85,0.001,3,1.10
INTEGRATION
8,4,2,1.2,.8,.5,.3,.15
1.0
SSW
2,0
5,0
6,80.5
0,0
ISSW
12,4
11,0
0,0
PRIM0      VALUES TO COMPARE WITH MBEM
53.6,-7.5,0,-7.5,60,0,0,0,95.5
PRIM1
-.008,0,0,0,-.0152,0,0,0,-.0274
SUBS
1
1,80000,.25,1,0
BENCH
1,-61.5,5,2,            1,1738.5,4805,2602
POINT A
2,31.4,5.7,-24.1,       2,1831.4,4805.7,2575.9
POINT B
3,-1000,-1000,2
REMOTE A
4,-1000,-1000,-24.1
REMOTE B
-99,0,0,0
END
```

Test data file : INTER.1

```
16,16
0,0,0
1,0,0,0
2,1,0,0
3,1,1.0,0
4,0,1.0,0
5,0,0,1
6,1,0,1
7,1,1.0,1
8,0,1.0,1
9,0,0,2
10,1,0,2
11 1,1.0,2
12,0,1.0,2
13,0,0,3
14,1,0,3
15,1,1,3
16,0,1,3
1,4,1,4,3,2
BOT 1
2,4,7,6,2,3
FRONT 1
3,4,8,7,3,4
RIGHT 1
4,4,5,8,4,1
BACK 1
5,4,5,1,2,6
LEFT 1
6,4,5,6,7,8
TOP 1
7,4,11,10,6,7
FRONT 2
8,4,12,11,7,8
RIGHT 2
9,4,12,8,5,9
BACK 2
10,4,9,5,6,10
LEFT 2
11,4,12,9,10,11
TOP 2
12,4,10,11,15,14
FRONT 3
13,4,11,12,16,15
RIGHT 3
14,4,9,13,16,12
BACK 3
15,4,14,13,9,10
LEFT 3
16,4,13,14,15,16
TOP 3
```

Test data file : INTER.2

```
 1, 5,1,1,1,1,0,2,0,0
 6, 6,1,7,2,1,2,2,0,0
 7,10,2,1,3,2,0,2,0,0
11,11,2,7,4,2,3,2,0,0
12,16,3,1,4,3,0,2,0,0
0,0,0,0,0,0,0,0,0,2,0,0
```

Test data file : INTER.3

```
HEADING
 3 SUBREGION PROBLEM
SPTRAC
16,16
1,9,0,0,-10
0,0
SUBS
3
1,10,.3,1
2,100,.25,1
3,1000,.25,1
BOUNDR
1,1,5
3,3,4
4,4,3
5,5,4
6,6,7
8,8,4
9,9,3
10,10,4
11,11,7
13,13,4
14,14,3
15,15,4
0,0,0
ITERATIONS
100,0.80,.001,4,1.05
INTEGRATION
10,4.4,2.4,1.6,1.1,0.4
100
SSW
6,.3
0,0
BENCH
1,.5,.5,.5,1
FIRST SUB
2,.5,.5,1.5,2
SECOND SUB
3,.5,.5,2.5,3
THIRD SUB
-99,0,0,0,0
END
```